International Organization of Research & Development (IORD)
ISSN: 2348-0831
Vol 04 Issue 02 | 2016

Structural and thermal analysis of Spring of Spring loaded safety valve using FEM

Mudassir Z Khan, Nitin A Kharche
Padm.Dr.V.B.Kolte College of Engineering Malkapur

ABSTRACT

This paper deals with the study of dynamic characteristics of spring of spring loaded relief valve and by using ANSYS Workbench. Mesh deformation due to the fluid- solid interaction between the valve disc and the surrounding fluid, are used to account for the motion of the spring for different Materials. The initial boundary conditions are applied to check the failures and then different materials are used with same boundary condition for evaluating the best suitable material. And then Analytical result are calculated.

Keywords— FEM, Blow down, Ansys, Relief valve

1. INTRODUCTION

The basic spring loaded pressure relief valve has been developed to meet the need for a simple, reliable system actuated device to provide overpressure protection. The spring load is adjustable to vary the pressure at which the Valve will open. When a pressure relief valve begins to lift the spring force increases. Thus system pressure must increase if lift is to continue. For this reason pressure relief valves are allowed an overpressure allowance to reach full lift. This allowable overpressure is generally 10% for valves on unfired systems. This margin is relatively small and some means must be provided to assist in the lift effort. Most pressure relief valves therefore have a secondary control chamber or huddling chamber to enhance lift. As the disc begins to lift fluid enters the control chamber exposing a larger area of the disc to system pressure .This causes an incremental change in force which overcompensates for the increase in spring force and causes the valve to open at a rapid rate. At the same time the direction of the fluid flow is reversed and the momentum effect resulting from the change in flow direction further enhances lift. These effects combine to allow the valve to achieve maximum lift and maximum flow within the allowable overpressure limits. Because of the larger disc area exposed to system pressure after the valve achieves lift the valve will not close until system pressure has been reduced to some level below the set pressure. The design of the control chamber determines where the closing point will occur. The difference between the set pressure and the closing point pressure is called blow down and is usually expressed as a percentage of set pressure.

The spring loaded pressure relief valve (also called direct operated relief valve) is a type of pressure relief valve (PRV) used to control or limit the pressure in a system or vessel. It performs a critical function in preventing excessive pressure and protecting the system and mechanical equipment. Due to its simple configuration spring loaded PRVs are widely used in lots of hydraulic systems.

2. OBJECTIVES OF WORK

The following are the objectives of the study:

- To investigate the problems occurs in the Safety Valve.
- To prepare 3D CAD model of Safety Valve geometry (Data Available from earlier research).
- To perform Finite element analysis of Safety Valve geometry with natural boundary conditions.
- To suggests the remedial actions, new material, and different shapes for Safety Valve geometry to solve the failures.

3. PROBLEM STATEMENT

To design Pressure Spring of spring loaded Safety Valve which can regulate the pressure in the system within given specified limit with regards to an axial and bending load by the flowing liquid'. Multiple objectives include material finalization, thickness requirement for restrictor plate, and stiffness finalization for spring. The geometric dimension should be such that, self-weight should be the operational parameter for the valve. Further FEA techniques are used to test this design to study the stress patterns and to ensure a durable design. Key constraints in designing the spring are geometrical parameters as well as operating parameters such as pressure and temperature. The spring used for the present work is spring of Lever Operated Spring Safety Valve and Analysis done on it.

4 .COMPUTER AIDED MODEL DESIGN OF SAFETY VALVES CAD/CAE

Software's used for design and analysis

- PRO/E – For 3D Component Design
- Pro/Assembly – For Assembling Components
- ANSYS Workbench – For CAE/FEM analysis

3D Modelling

The essential difference between Pro/ ENGINEER and traditional CAD systems is that models created in Pro/ENGINEER exist as three-dimensional solids. Other 3D modellers represent only the surface boundaries of the model. Pro/ENGINEER models the complete solid. This not only facilitates the creation of realistic geometry but also allows for accurate model calculations such as those for mass properties.

Parametric Design

Dimensions such as angle, distance, and diameter control Pro/ENGINEER model geometry. You can create relationships that allow parameters to be automatically calculated based on the value of other parameters. When you modify the dimensions, the entire model geometry can update according to the relations you created.

Feature-Based Modelling

You create models in Pro/ENGINEER by building features. These features have intelligence, in that they contain knowledge of their environment and adapt predictably to change. Each features asks the user for specific information based on the feature type. For example, a hole has a diameter, depth and placement while a round has a radius and edges to round.

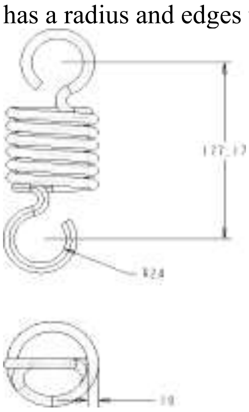

Fig.1 Dimension of spring

Fig.2 Design Model of Spring

5. FINITE ELEMENT ANALYSIS OF SPRING OF SAFETY VALVES

The finite element method (FEM), sometimes referred to as finite element analysis (FEA), is a computational technique used to obtain approximate solutions of boundary value problems in engineering. Simply stated, a boundary value problem is a mathematical problem in which one or more dependent variables must satisfy a differential equation everywhere within a known domain of independent variables and satisfy specific conditions on the boundary of the domain. Boundary value problems are also sometimes called field problems. The field is the domain of interest and most often represents a physical structure.

General procedure for finite element analysis

The Finite Element Method (FEM) is a numerical technique used for getting approximate solutions of partial differential equations. Finite Element Analysis is a simulation technique which is used to find out the behaviour of components, structures and equipment for various loading conditions such as applied forces, pressures and temperatures. Therefore a complex engineering problem with non-standard shape and geometry can be solved using finite element analysis. The finite element analysis methods result in the stress distribution, displacements and reaction loads at supports etc. for the model.

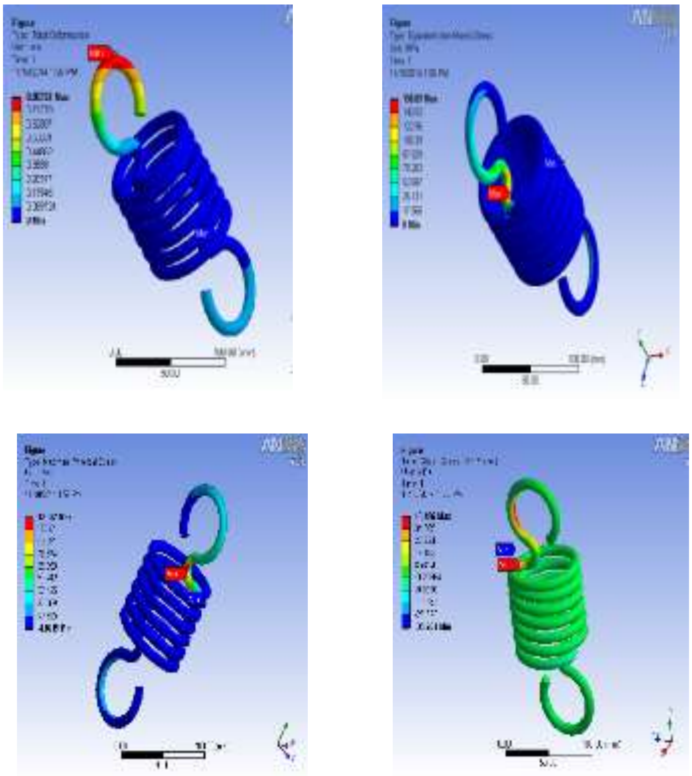

Fig.3 ansys result for Existing Material

The various steps involved in the finite element analysis are –

- Select suitable field variables and the elements.
- Discretise the Domain
- Select shape functions.
- Find the element properties.
- Assemble element properties to form global properties.
- Impose the boundary conditions.
- Solve the matrix equations to get the nodal unknowns.
- Make the additional calculations to get the required values.

Finite element analysis of spring using Existing Material

Structural steel/Alloy steel are used as an existing materials. By using the existing material the analysis is performed in Ansys.

Material Property of Structural steel are

Young's Modulus (MPa)	1750000
Poisson's Ratio	0.25
Density (kg/mm3)	5.74E-06

Table 1: Material Property of Structural steel

Young's Modulus (MPa)	200000
Poisson's Ratio	0.3
Density (kg/mm3)	7.85E-06

Finite element analysis of spring using La2 zr2 o7 alloy (Lanthanum zirconium oxide) Material
Second Material used for the analysis of safety valve is La2 zr2 o7 alloy. The Material Properties are as follow:

$$k = \frac{1}{4c-4} + \frac{0.615}{5}$$

$4c -$

Spring Index

$$C = \frac{Mean\ diameter\ of\ Coil}{Wire\ diameter}$$

Maximum shear stress is $\tau =$

$54.46\ Mpa$ Maximum deflection of spring due to applied load

$$\Delta = \frac{8.w.c^3.n}{G.d}$$

- Using Titanium alloy G =42*E03

  $\Delta = 8.064\ mm$

- Using la2zr2O7 G =43*E03

  $\Delta = 7.87\ mm$

• Using Steel & magnesium alloy G =16.5*E03
$\Delta = 20.52\ mm$

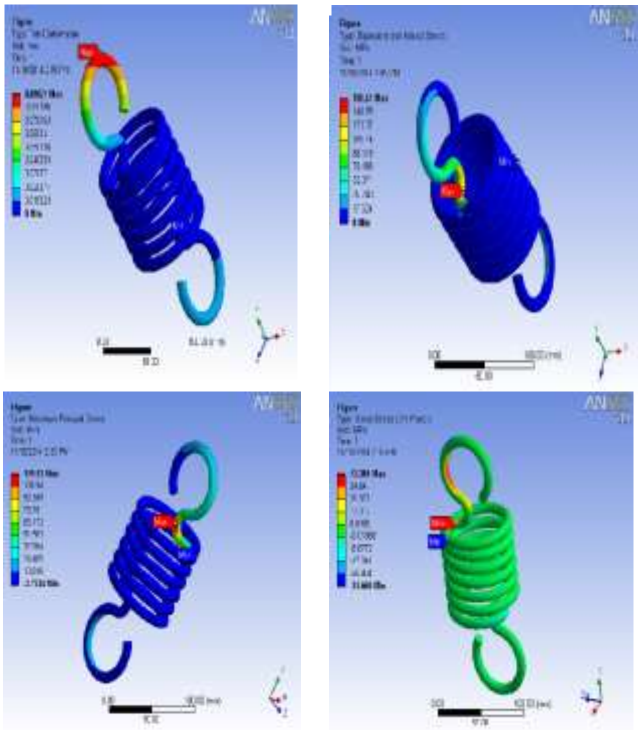

Fig.4 ansys result for La2 zr2 o7 Material

Similar Analysis can be done by using the different material and compared with existing material.

6. ANALYTICAL CALCULATIONS

Maximum applied Pressure (p) =0.2Mpa=0.2N/mm^2
Diameter of valve seat = 42mm Max Tensile force acting on spring

$$W = \frac{\pi}{4}(D^2) * p$$

Maximum Shear stress due to applied pressure

$$\tau = k * \frac{8.w.c}{\pi.d^2}$$

K- Wahl's Stress factor

7. RESULT

	Existing Material	Magnesium Alloy Material	Aluminum Silicate Alloy Material	Titanium Alloy	La2 Zr2 O7
Total Deformation	0.80752 mm	1.7945 mm	1.7945 mm	1.7262 mm	9.021e-002 mm
Equivalent Stress	158.09 MPa	158.09 Mpa	158.09 Mpa	157.71 Mpa	158.61 Mpa
Maximum Principal Stress	120.32 MPa	120.32 Mpa	120.32 Mpa	124.45 Mpa	119.53 Mpa
Shear Stress	43.406 Mpa	43.406 Mpa	43.406 Mpa	43.629 Mpa	43.304 Mpa
Strain Energy Lost due to Heat	0.935	2.522	4.142	1.9438	0.10415 mJ

Result obtained after the analysis of spring loaded safety valve are shown in following Table

8. CONCLUSION

From above Study it is conclude that for spring of spring loaded safety valve La2 Zr2 O7 is best material than existing material as per the Ansys result obtained and analytical calculations. In order to examine the performance characteristics of the safety valve of high-pressure, Structural and thermal analysis were conducted. The following conclusion was obtained.

REFERENCES

[1]. Arend Beune (2009), "Analysis of HighPressure Safety Valves", A Catalogue Record is Available from the Eindhoven University of Technology Library, ISBN: 978- 90-386-2006-0.
[2]. Beune A, Kuerten J G M and van Heumen M P C (2012), "CFD Analysis with FluidStructure Interaction of Opening HighPressure Safety Valves", Journal of Science Direct Computers & Fluids, Vol. 64, pp. 108-116.
[3]. Gabor Licsko, Alan Champneys and Csaba Hose (2009), "Nonlinear Analysis of a Single Stage Pressure Relief Valve", International Journal of Applied Mathematics (IJAM), Vol. 39, No. 4, p. 12.
[4]. Kishan Patel (2014), "CFD Analysis of A Pressure Relief Valve", International Journal for Scientific Research & Development, Vol. 2, No. 04, ISSN: 23210613.
[5]. Mehul S Rajapati and Raut D N (2010), "Analysis of Pressure Relief Valve Using Matlab Simulink", 37th National and 4th International Conference on Fluid Mechanics and Fluid Power, December, IIT Madras, Chennai, India.
[6]. Rathod V D, Kada G A and Patil V G (2014), "Design and Analysis of Pressure Safety Releas Valve by Using Finite Element Analysis",International Journal of Engineering Trends and Technology (IJETT), Vol. 13, No. 1, ISSN: 22315381.
[7]. Xue Guan Song, Lin Wang and Young Chul Park (2010), "Transient Analysis of Spring Loaded Pressure Relief Valve Using CFD", Journal of Pressure Vessel Technology, Vol. 132.
[8]. Xue Guan Song, JiHoon Jung, Hyeong Seok Lee, Dong Kwan Kim and Young Chul Park (2010), "2D Analysis of Pressure Relief Valve by CFD", 9th WSEAS Int. Conference on Applied Computer and Applied Computational Science, ISSN: 1790-5117, ISBN: 978960-474-173-1.