Shape Optimization of Double Ended Shear Beam Load cell For Volume and Sensitivity

A.V. Sutar\textsuperscript{1} Aditya D. Patil\textsuperscript{2} Pratik H. Koli\textsuperscript{3} Shankar S. Pote\textsuperscript{4}

\textsuperscript{1}Assistant Professor, \textsuperscript{2,3,4}B.E. Student

Department of Mechanical Engineering

\textsuperscript{1,2,3,4}D.K.T.E.’s Textile and Engineering Institute, Ichalkaranji, Dist: Kolhapur – 416115, India

Abstract— In this work ‘Double Ended Shear Beam’ type load cell is considered for shape optimization by using finite element method. The shape optimization is carried out to minimize the weight of ‘Double Ended Shear Beam’ type load cell without exceeding allowable strain. The intention of the work is to create the geometry of ‘Double Ended Shear Beam’ type load cell to find out the optimum solution. FEM software ANSYS 14.5 is used for shape optimization of ‘Double Ended Shear Beam’ type load cell. If the stress & strain values are within the permissible range, then certain dimensions are modified to reduce the amount of material needed. Experimental verification is carried out by photoelasticity technique with the help of Polariscope Instrument. Using Polariscope the stress distribution pattern is verified for a photoelastic model of ‘Double Ended Shear Beam’ using Diffused light Polariscope set up. Experimental results are compared with FEM results.

Key words: Load cell, optimization, stress, strain, FEM, Sensitivity, Photoelasticity

I. INTRODUCTION

Load cells have long been used to sense and measure force and torque. When properly designed and used they are very accurate and reliable sensors. Load cells are applied in several different fields, usually for weighing Measurements. Among many other things, food, vehicles, and animals are weighed daily with load cells. The gripper of a robotic arm that picks up an object can be equipped with load cells in order to provide compression force feedback to the control system to prevent the object from being damaged or released too early. Also, load cells can be used to sense the compression forces during a robot’s walk to provide data for the equilibrium-controlling system. In industrial machinery, rods, beams, wheels and bars are instrumented in order to check the forces exerted on them. The volume or level of a tank can be measured indirectly by means of a load cell that monitors the total weight. Lift units can also have a load’s total weight measured to prevent overload. Because of such a variety of possible applications, load cells are very important.

A load cell is a transducer that is used to convert a force into electrical signal. This is indirect conversion which occurs in two stages. Through a mechanical arrangement, the force being sensed deforms a strain gauge. The strain gauge converts the deformation (strain) to electrical signals. Most industrial measurements of force are made with strain gauge load cell. The majority of today’s designs use strain gauges as the sensing element, either foil or semiconductor. Foil gauges offer the largest choice of different types and in consequence tend to be the most used in load cell designs. Strain gauge patterns offer measurement of tension, compression and shear forces. A vast number of load cell types have developed over the years; the previous designs simply used a strain gauge to measure the direct stress which is introduced into a metal element when it is subjected to a tensile or compressive force. A bending beam type design uses strain gauges to monitor the stress in the sensing element when subjected to a bending force. Load cell provides their output under tension or compression.

II. TAGUCHI METHOD

The Taguchi method involves reducing the variation in a process through robust design of experiments. The overall objective of the method is to produce high quality product at low cost to the manufacturer. The Taguchi method was developed by Dr. Genichi Taguchi of Japan who maintained that variation. Taguchi developed a method for designing experiments to investigate how different parameters affect the mean and variance of a process performance characteristic that defines how well the process is functioning. The experimental design proposed by Taguchi involves using orthogonal arrays to organize the parameters affecting the process and the levels at which they should be varies. Instead of having to test all possible combinations like the factorial design, the Taguchi method tests pairs of combinations. This allows for the collection of the necessary data to determine which factors most affect product quality with a minimum amount of experimentation, thus saving time and resources.

The Taguchi method is best used when there is an intermediate number of a variable (3 to 50), few interactions between variables, and when only a few variables contribute significantly.

Taguchi specified three situations for selection of optimum parameters:

1) Larger the better.
2) Smaller the better.
3) On target, minimum variation.

Taguchi method of Design of Experiment:

1) Define the process objective, more specifically, a target value for a performance measure of the process.
2) Determine the design parameter affecting the process. Parameters are the variables within the process that affect the performance measure such as temperature, pressure, etc.
3) Create orthogonal array for the parameter design indicating the number of conditions for each experiment.
4) Conduct the experiments indicated in the completed array to collect data on the effect on the performance measure.

Critical parameters of load cell defined for optimization of load cell and determination of orthogonal array as below:
Shape Optimization of Double Ended Shear Beam Load cell For Volume and Sensitivity

(IJSRD/Vol. 5/Issue 01/2017/230)

All rights reserved by www.ijsrd.com
847

A. Length
Fig. 1:

B. Width
Fig. 2:

C. Height
Fig. 3:

Length, Width and Height, three parameter which are affect the performance of double ended shear beam type load cell.

<table>
<thead>
<tr>
<th>Levels</th>
<th>Parameters</th>
<th>A</th>
<th>B</th>
<th>C</th>
</tr>
</thead>
<tbody>
<tr>
<td>Length</td>
<td></td>
<td>208</td>
<td>260</td>
<td>312</td>
</tr>
<tr>
<td>Width</td>
<td></td>
<td>49.6</td>
<td>62</td>
<td>74.4</td>
</tr>
<tr>
<td>Height</td>
<td></td>
<td>60</td>
<td>75</td>
<td>90</td>
</tr>
</tbody>
</table>

Table 1: Taguchi design factors and their levels

<table>
<thead>
<tr>
<th>Sr. No.</th>
<th>A</th>
<th>B</th>
<th>C</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>208</td>
<td>49.6</td>
<td>60</td>
</tr>
<tr>
<td>2</td>
<td>208</td>
<td>62</td>
<td>75</td>
</tr>
<tr>
<td>3</td>
<td>208</td>
<td>74.4</td>
<td>90</td>
</tr>
<tr>
<td>4</td>
<td>260</td>
<td>49.6</td>
<td>75</td>
</tr>
<tr>
<td>5</td>
<td>260</td>
<td>62</td>
<td>90</td>
</tr>
<tr>
<td>6</td>
<td>260</td>
<td>74.4</td>
<td>60</td>
</tr>
<tr>
<td>7</td>
<td>312</td>
<td>49.6</td>
<td>90</td>
</tr>
<tr>
<td>8</td>
<td>312</td>
<td>62</td>
<td>60</td>
</tr>
<tr>
<td>9</td>
<td>312</td>
<td>74.4</td>
<td>75</td>
</tr>
</tbody>
</table>

Table 2: Experiments of Taguchi

To analyze design of taguchi experiments Minitab software is used to find out the experiment from taguchi table, which is optimum and follows the larger, is better low. The following graph is generated by Minitab software:

Fig. 4: Graph of main effect plot for means (vol. v/s values)

Graphs shows smaller is better value in which Y axis is output in terms of volume and X axis is dimensions of the model.

In graphs smaller value of volume is at length 208 mm and height 60 mm and width 49.6 mm from the table we taken these value for optimum design.

III. FINITE ELEMENT ANALYSIS

Finite Element Analysis is an engineering analysis technique that is utilized to simulate the behavior of complex structures for which no exact solutions exist. While it has its historical beginnings in structural analysis, finite element analysis is now applied to a wide variety of engineering areas, including fluid flow, heat transfer, and electromagnetic. The basic premise of finite element analysis is quite simple: if a problem is too complex to solve, create smaller, more manageable subsets for which you can find solutions. Then, going on the assumption that the behavior of the complex whole can be approximated by the sum of the behaviors of its parts, assemble all the individual results of the separate elements into one solution for the overall problem.

The process creates a mathematical model that subdivides a complex, and often large, model into small regular-shaped subsets, called elements. The behavior of the elements and their interaction with one another is observed and analysed at a finite number of connecting points, located at the element edges and vertices. These special points are called nodes. The network of nodes and elements is the mesh. The finite element process uses the geometric and material properties of the elements, and the effects of known loads and restraints acting on the body, to calculate a characteristic response at the nodes. In structural analysis, this response is nodal displacement. The secondary values, which in structural analysis are the stresses and strains, are all calculated from the primary results.
Stress simulation performs a particular type of finite element analysis called Linear Static Analysis. The various steps involved in stress/strain simulation (Fig. No. 4) are:

- **Create a simulation model:**
  To create a simulation model use solid modeling software like AutoCAD, CATIA, etc. One can easily create solid model using these softwares. Import solid model into the analysis software like ANSYS 14.5.

- **Select Modeling Units:**
  Now we can select units for model which are generally in SI, CGS and English. Also we can set custom units for individual model.

- **Generate finite element mesh:**
  FEA software automatically creates a mesh on the solid models with default element sizes based on geometry's dimensions. We can change the element size according to the required accuracy.

- **Define material properties:**
  FEA software provides the standard material library. We can also add the new materials and also we can enter their properties in to the library.

- **Specify load and restraints:**
  Loads and restraints applied on the model simulate its real life working environment.

- **Submit model to analysis:**
  In this step, we can submit the model to the software to analyze. FEA software itself analyses and gives required results.

- **Review the result:**
  It gives or evaluates the results stress and deformation contour plots, strain and deformation contour plots, deformed and undeformed shapes, mode shape displays, animation and other visual and numeric outputs.

After analysis in ANSYS, stress and strain values are obtained. Load cell has maximum strain 261.93 µstrain and optimized load cell has maximum strain 396.54 µstrain.

Also critical parameters changes, length 260 mm to 208 mm, width 62 mm to 49.6 mm and height 75 mm to 60 mm.

<table>
<thead>
<tr>
<th>Type</th>
<th>Volume (mm³)</th>
<th>Maximum strain (µstrain)</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>B</td>
<td>C</td>
</tr>
<tr>
<td></td>
<td>X 10⁶</td>
<td></td>
</tr>
</tbody>
</table>
### IV. Photoelastic Stress Analysis

Photoelasticity is an experimental method to determine the stress distribution in a material. The method is mostly used in cases where mathematical methods become quite cumbersome. Unlike the analytical methods of stress determination, photoelasticity gives a fairly accurate picture of stress distribution, even around abrupt discontinuities in materials. The method is an important tool for determining critical stress points in a material, and is used for determining stress concentration in irregular geometries. For finding stress, polariscope is used, generally two types of polariscope used, 

- Plane Polariscope
- Circular polariscope

The polarized light when passed through the stressed transparent model, an interference pattern on fringes is formed. With the help of Photoelastic stress analysis, it is possible to determine the value of principal stress difference (\(\sigma_1 - \sigma_2\)) at any point in the model. The stress fringe value is needed to determine the stresses in photo-elastic model.

The photoelastic model of ‘Double Ended Shear Beam’ load cell is developed as follows. Cy 230 araldite along with hardener Hy 951 is used for preparation of photoelastic model. 100 parts of araldite mixed with 9 parts of hardener by weight is used for casting the sheets. The model was carefully hand finished, so that no residual stresses are present in it. To determine values for stress-optic coefficient and fringe order for the material used, the general procedure is, to apply increasing load and identify the fringe order and stress at specified fringe numbers.

### Tardy Compensation:

Tardy compensation technique is used to determine the value of stresses. In this method, a plane polarizer (quarter-wave plates removed physically or optically) is first used to find the directions of the principal stresses at any selected point. The polarizer/analyzer pairs are then rotated as a unit until their axes are aligned with the bi-refringent indcial axes, and the quarter plates are then reinserted with their axes oriented at the usual 45-degree angle with respect to the new polarizer axes. The system is now in the standard circular dark-field polariscope condition. Up to this point the polarizer/analyzer axes have been kept orthogonal to each other, but now the analyzer is rotated separately until one of the fringes moves over the selected point. In rotating the analyzer up to 90 degrees, the polariscope is changing from a dark-field configuration to a light-field configuration. This change will cause a fringe to move a half order for rotation \(\pi\), less than 90º, the ratio, \(\pi/90\) will correspond to a fractional fringe change relative to a half-order. Thus I the angle through which the analyzer has been assigned to the selected point. Many polariscopes index a half-circle scale in terms of 100 divisions, so that the fractional order can be read off directly from the scale.

At the free boundaries one of the principal stresses, which is normal to boundary is zero. Therefore the value of tangential stresses, say \(\sigma_1\), \((\sigma_2\text{being zero})\) at free boundary is given by,

\[
\sigma_1 = \frac{Nf\sigma}{h} \quad \text{(2)}
\]

Where, \(N\) = Fringe order
\(f\) = Stress fringe value (N/m)

\(h\) = Model thickness (m)

Using equation (2) stresses at different positions are determined. Those values are as listed in Table No.3

<table>
<thead>
<tr>
<th>Load (kg)</th>
<th>Fringe Order</th>
<th>Stress (N/mm²)</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>1.48</td>
<td>36.61</td>
</tr>
<tr>
<td>7</td>
<td>2</td>
<td>49.48</td>
</tr>
</tbody>
</table>

### Scaling model results to prototype:

By knowing stresses in photo-elastic model it is possible to determine the stresses in prototype, say a machine component. The model is geometrically similar to the prototype and is loaded in the similar manner as prototype. Material used for the prototype is usually metal which is different from the model material. Similarly size of the prototype is also different from that of the model. Even with these, the determination of stresses in prototype from stresses in model is simple. This is because the material constants \(E\) and \(\mu\) do not affect the stress distribution. The stress \(\sigma\) at any point in body depends upon the load \(P\), thickness ‘\(h\)’ and typical linear dimension. Let \(\sigma_m\), \(h_m\) and \(\sigma_p\), \(h_p\) denote the stress, thickness and linear dimensions for model and prototype respectively.

\[
\frac{\sigma_m}{\sigma_p} = \left(\frac{h_p}{h_m}\right)^3 \quad \text{(3)}
\]

Using above equation we can calculate the stress induced in the prototype as listed below,

<table>
<thead>
<tr>
<th>Load (kg)</th>
<th>(\sigma_p) (N/mm²)</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>31.40</td>
</tr>
<tr>
<td>7</td>
<td>44.25</td>
</tr>
</tbody>
</table>

### Table No. 5 gives the comparison of Stress Values obtained by ANSYS (\(\sigma_p\)) and Photoelasticity (\(\sigma_p\))

<table>
<thead>
<tr>
<th>Load (kg)</th>
<th>(\sigma_p) (N/mm²)</th>
<th>(\sigma_p) (N/mm²)</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>31.40</td>
<td>36.61</td>
</tr>
<tr>
<td>7</td>
<td>44.25</td>
<td>49.48</td>
</tr>
</tbody>
</table>

### Table 5: Maximum Stress Value (\(\sigma_m\)) and (\(\sigma_p\))

Both values are very close to each other. We can calculate percentage error.

\[
\%\text{Error} = \frac{\sigma_p - \sigma_m}{\sigma_m} \times 100 \quad \text{(4)}
\]

\(\sigma_p\) = Stress determined using ANSYS Software.
\(\sigma_m\) = Stress determined using Photo-Elasticity Technique.

Calculated values are,
For 5 kg load, Error = 14.23%
For 7 kg load Error = 10.56%

V. CONCLUSION

This work presents shape optimization using finite element method. Using FEA software ANSYS directly 3-D discretised model of 'Double Ended Beam' type Load Cell is build with solid elements. Further the optimization of model is done for reduction of weight for given criteria that maximum strain induced should be below 1000 µ strain.

The length of load cell is reduced from 260 to 208 mm. Height is reduced from 75 to 60 mm and width is reduced from the 62 to 49.6. The overall volume of load cell is reduced from 1.0484 x 106 mm3 to 0.53705 x 106 mm3.

The sensitivity observed for original load cell was 0.001335 µ strain/N. The sensitivity observed for optimized load cell was 0.002022 µ strain/N. Which increases by 33.97%.

By observing results obtained by photo-elasticity or FEM it can be seen that results of photo-elasticity is slightly varying from FEM by minimum error 14.23 %. Reasons behind this error is some of the errors like initial double refraction in the model, error due to machining, other internal stresses, lack of dimensional accuracy, faulty loading, less sensitive electronic equipment, and change in meshing size.

REFERENCE