Available online www.jsaer.com

***Journal of Scientific and Engineering Research*, 2017, 4(5):157-161**

Research Article

ISSN: 2394-2630
CODEN(USA): JSERBR

Evaluation of Stress Concentration Factor in Steel Elastic Element of Load Cell

Abubaker M. Saeed Hamed, Mohammad Qasim Abdullah

Mechanical Department, Engineering College – Baghdad University, Iraq

Abstract This paper study the stress concentration in one of the important force measuring device (load cell) [1]. The elastic element of load cell have many type but in this paper We will be interested in studying one of the most common types of load cell, we will study the stress concentration [2-4] in one popular spring element design is the binocular configuration, which is a beam with two holes and a web of beam material removed. Therefore we determined the stress concentration in tow design of binocular load cell type using ansys [5] simulation program.

Keywords Load cell, stress distribution in load cell , load cell deformation , load cell concentration factor

Introduction

Most industrial measurements of force are made withstrain gauge load cell. It utilizes strain gauges bonded to steel body to give an electrical output proportional to applied force. Different types of load cells are beam type, 'S' type, cantilever type etc. These are used for tension measurements, weighing, industrial automation and research applications, etc.

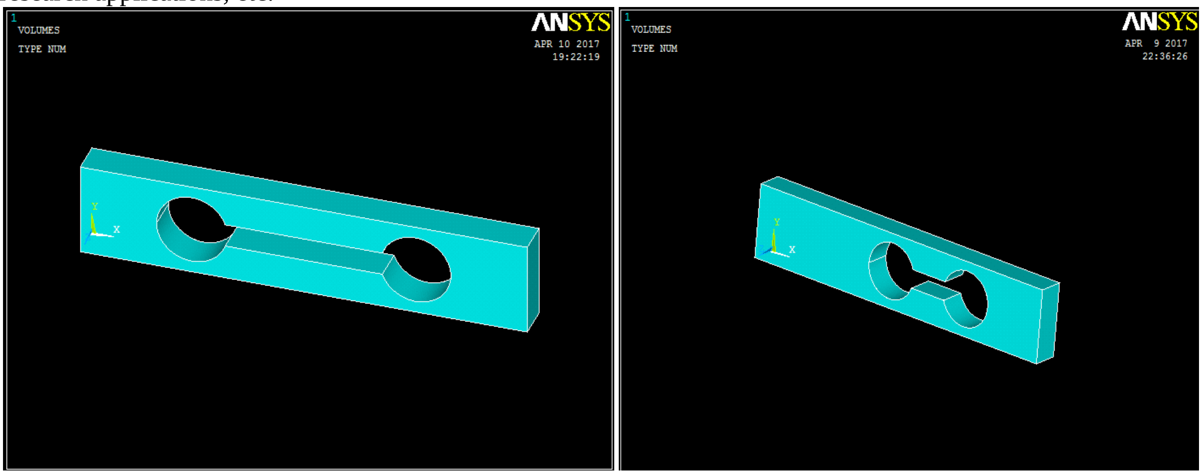

Figure 1: The two model of binocular load cell

Most commonly, shape optimization is the process of changing the physical dimensions of a structural part to reduce weight while staying within design constraints, usually maximum stress or deflection shape optimization can include, but is not limited to optimizing items such as fillet radius, hole diameter, and width or height of a part. Usually, shape optimization is performed on solid models, although there are many applications for surface models (shells) (ANSYS 6.1, Theory Reference, Design Optimization).

Shape optimization links finite element analysis and parametric geometric construction to make it possible to explore different design options. It involves varying certain design parameters (e.g. dimensions) while observing other criteria (e.g. maximum allowable stress) until the best and most economical design is found. While finite element analysis is primarily a tool used to verify a design, it can also be the first step toward improving a design. A first-pass finite element analysis might show that stresses and other design responses are well below a prescribed limit, which indicates that it is safe to make some design changes. For example, you may modify certain dimensions to reduce the amount of material needed.

Model Verification

The two model made from steel with young modules equal to (200e9) and all dimensions of the tow to model are show in figure (2)

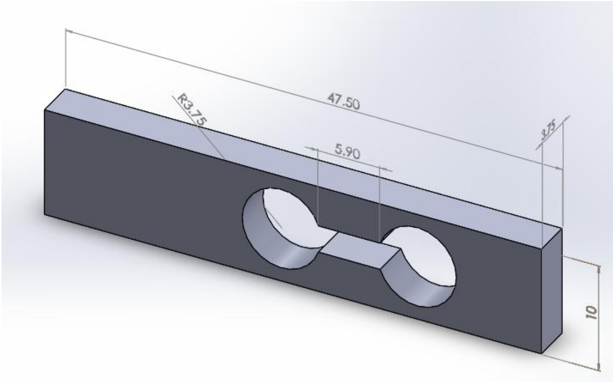

Figure 2: model 1

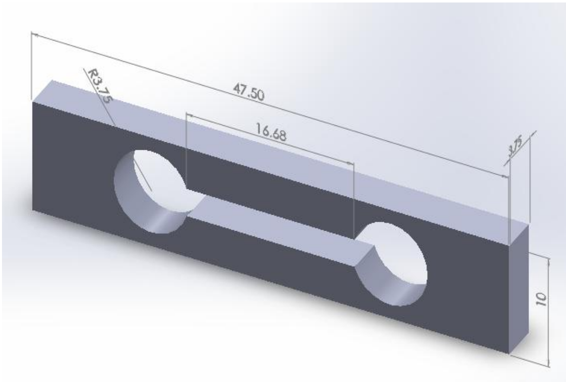

Figure 3: Model 2

Finite Element Method

Finite Element [3] 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.

CREATE A SIMULATION MODEL

↓

SELECT MODELING UNITS

↓

SPECIFY THE LOADS & RESTRAINTS

↓

GENERATE THE FINITE ELEMENT MESH

↓

DEFINE THE MATERIAL PROPERTIES

↓

SUBMIT MODEL TO ANALYSIS

↓

REVIEW THE RESULTS

Figure 4: Block diagram for simulation process

Results and Analysis

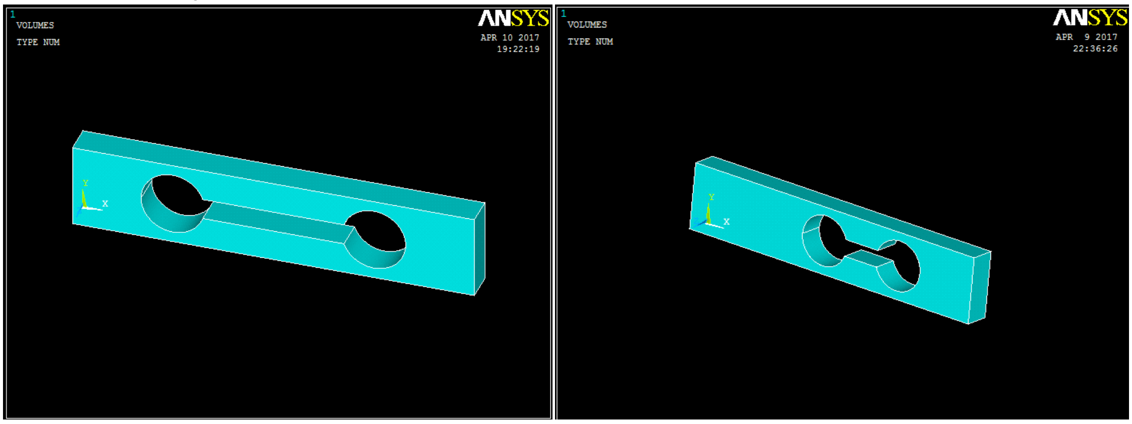

Figure: 5 Tow model of 'binocular' Type Load Cell 3-D Model

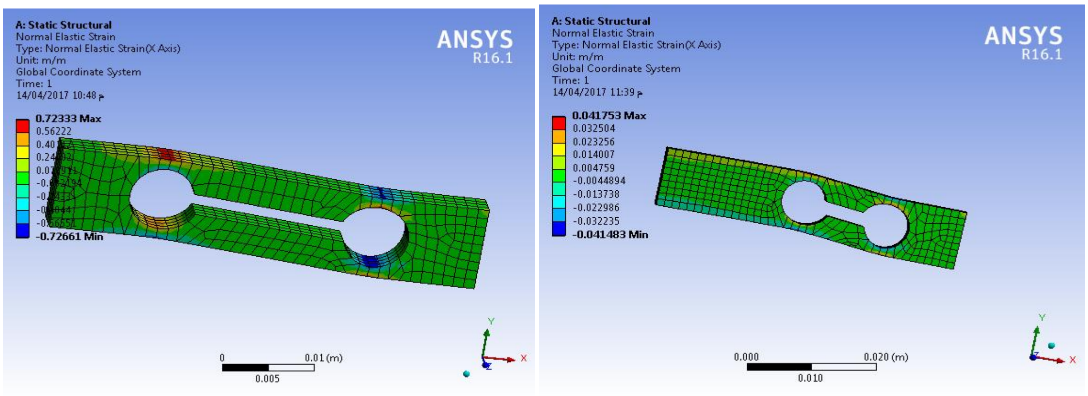

Figure 6: Strain distribution

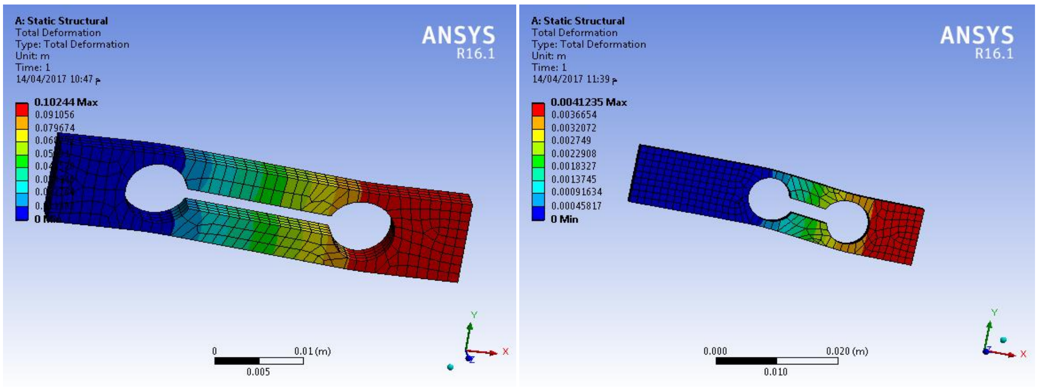

Figure 7: Deformation of the models

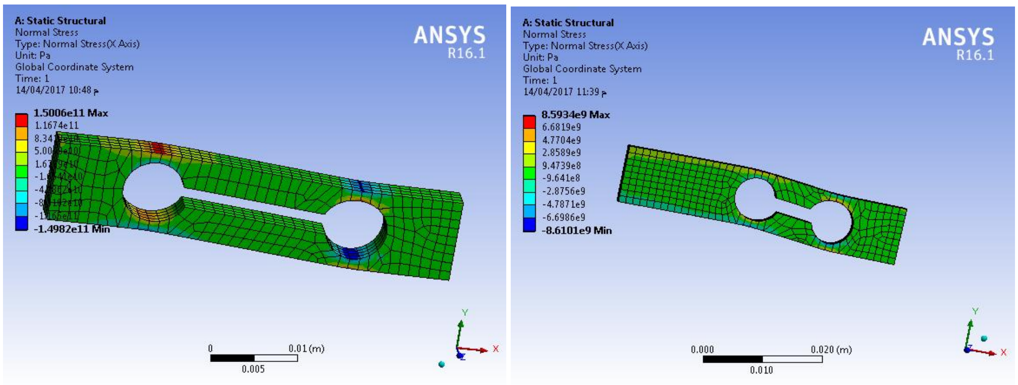

Figure 8: Stress distribution

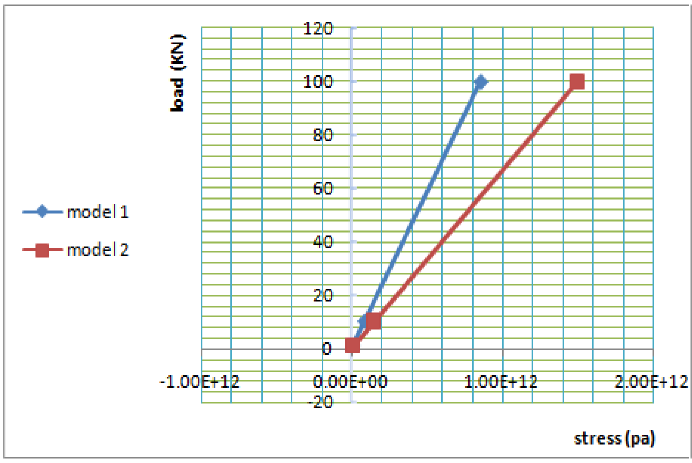

Figure 9: Load – stress relationship

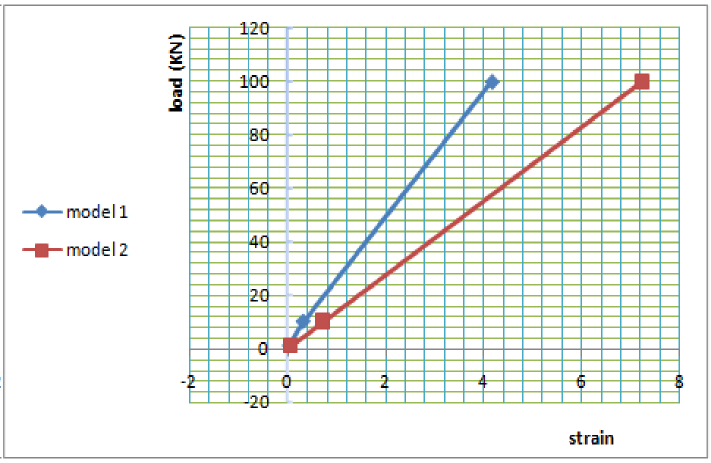

Figure 10: Load – strain relationship

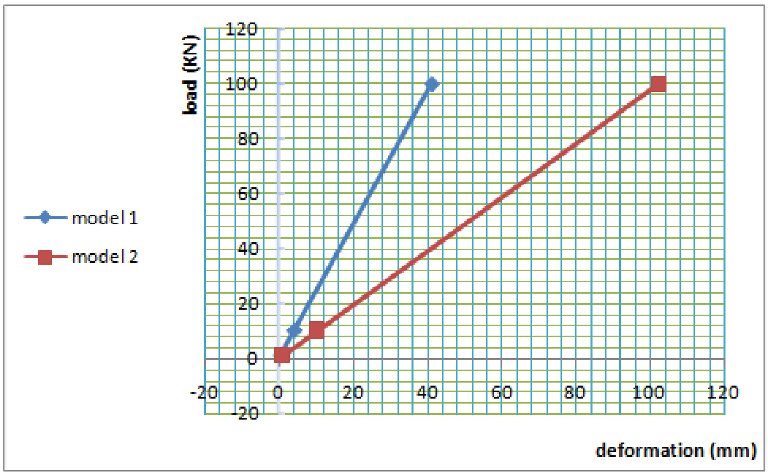

Figure 11: Load – deformation relationship

Table 1: The Results

	First model			Second model		
Load(KN)	Max. stress (pa)	Max. strain	Max. deformation (mm)	Max. strain	Max. stress(pa)	Max. deformation (mm)
1	8.593e9	0.04175	0.4123	0.07233	1.5006e10	1.0244
10	8.5934e10	0.32504	4.1235	0.7233	1.5006e11	10.244
100	8.5934e11	4.1753	41.235	7.233	1.5006e12	102.44

Nominal Stress

$Kt=\frac{\sigma max}{\sigma n}$

$\sigma n=\frac{MY}{I}$ $Y=h/2$ $I=bh^3/12$ $M=PL$

for first model : h=2.5mm ; b=3.75; L=25 mm

for the second module : h=2.5mm ; b=3.75; L=40 mm

Table 2: The results of Kt

Model	σmax(pa)	σn (pa)	Kt
1	8.593e9	6.4e9	1.3426
2	1.5006e10	1.024e10	1.4654

As we see that the stress, strain and deformation in the second model is great than the first model that mean the second model has a great sensitivity due to change in load rather than the first.

Conclusion

Load cell design is challenging due to the complex geometry of spring elements. ansys solid models are useful for predicting strain in these designs, for locating strain gage mounting positions and especially for optimizing maximum strain for the desired load range.

As we see in the result of analysis of to model if we need high accuracy in measurement of the force we most use the second model but the problem in this model that the stress construction is very high for that we can't use this model for high force. In the other hand if we need to measurement high force we can use the first module but we will get error in measured the load more than in the first one due to differences in strain of both models stress concentration factor have direct proportional with the sensitivity of load cell

Reference

[1]. Vishay Micro-Measurements “Model P3 Strain Indicator And Recorder” Instruction Manual.4, Measurements Group, Raleigh, NC (2005).
[2]. Ahsan, R. U., Prachurja, P., Ali, A. R. M., and Mamun, M. A. H., “Determination of effect of elliptic notches and grooves on stress concentration factors on notched bar in tension and grooved shaft under torsion”, Journal of Naval Architecture and Marine Engineering, Vol.10 Issue 1, 2013.
[3]. A. Kamble Vijay and P. N. Gore " Use of FEM and photo elasticity for shape optimization of S type load cell " Indian Journal of Science and Technology
[4]. Muhsin J. Jweeg, Mohammad Qasim Abdullah, Issam K. Jasim "Effect of Design Parameters on The Location of The Weakest Section and Stress Concentration of Spur Gear Tooth" Fifth Scientific Conference of College of Engineering, University of Baghdad.
[5]. ANSYS 6.1, Theory Reference, Design Optimization.