© 2019 IJSRSET | Volume 6 | Issue 2 | Print ISSN: 2395-1990 | Online ISSN : 2394-4099
Themed Section : Engineering and Technology
DOI : 10.32628/ IJSRSET1963564

Review on Method of Optimize the Deflection of Beams by Conducting Bending Test Using ANSYS Workbench 15

Mulayam Kumar [1], Dr. Simant[2], Vijay Gupta[3]
PG Scholar[1], Director at S.I.T.M B.B.K[2], Assistant Professor[3],
[1,2]Department of Mechanical Engineering, Sagar Institute of Technology and Management
[3]Department of Mechanical Engineering, B.B.D.N.I.T.M, Lucknow, Barabanki, Uttar Pradesh, India

ABSTRACT

The present work has been carried out to study the effect of the varying the load at different materials (Aluminium Alloy 7075-T6, stainless steel 305 and Structural Steel 345w) on deflection. The simply supported beam has been subjected to varying load 5000N – 10000N and cantilever beam has been subjected to varying load 500N-1000N. The result obtained is in form of Directional Deflection and Equivalent Stresses. This analysis is done by the ANSYS Workbench 15.0 software under the static structural analysis further this result has been optimized using TAGUCHI METHOD using MINITAB17.

Keywords : ANSYS Workbench , TAGUCHI METHOD, MINITAB17, FEM

I. INTRODUCTION

Beam is a half mast or horizontal structural limb casing a gap among one or additional supports, & carrying vertical loads across (transverse to) its longitudinal axis, as a purlin, girder or rafter. Three main kinds of beams are:

- Simple span , supported at both ends
- Continuous , supported at more than two points
- Cantilever , supported at one end with the other e overhanging & free.

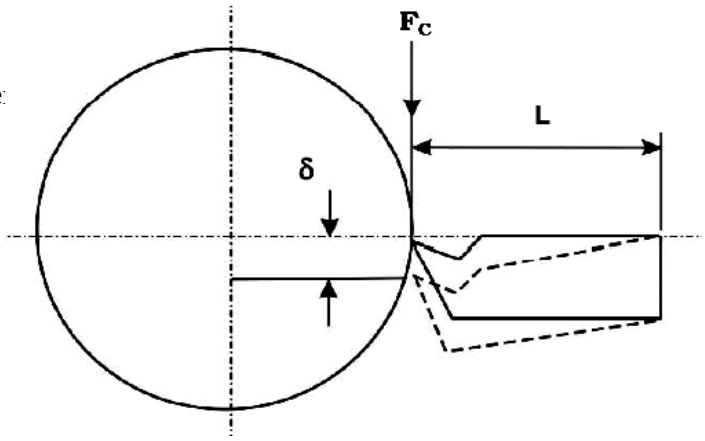
DEFLECTION OF THE CUTTING TOOL DUE TO THE CUTTING FORCE(S):

Under the work of the cutting force , call **Fc** in turning, the tool or tool holder elastically deflects as shown in Fig.1 Such tool deflection, δ is proportional to the magnitude of the cutting force, **Fc**, like as,

Correction the tool as a cantilever beam, we can write the deflection of the tool by following way

$$\delta = Fc\left(\frac{L\text{^}3}{3EL}\right)$$

Since for a cutting tool & its holder, E & I are fixed so we can write, **δ α Fc**



INTRODUCTION OF ANSYS:

In today world, where time is more influential in project planning & iterative calculations. Prototype experiments charges time, money, accuracy & effort of many intellectuals. Simulations or computer aided design comes in to sort these things up. ANSYS is one those tools which are currently used with a lot of applications which have significant impacts on our daily life founded in 1970 by Dr. John A. Swanson as Swanson Analysis Systems, Inc. SASI. Its primary goal was to develop & market finite element analysis software for structural physics that could simulate –

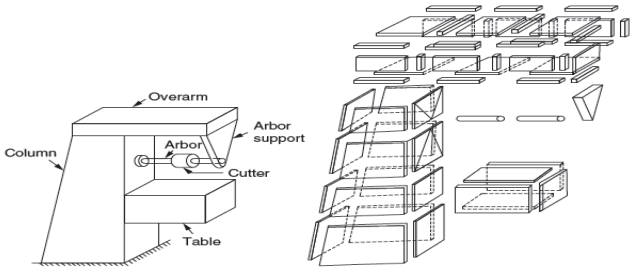
- Static (stationary)
- Dynamic (moving)
- Thermal (heat transfer) problems.

SASI developed its business in parallel with the growth in computer technology & engineering needs. The company grew by 10 percent to 20 percent each year, & in 1994 it was sold to TA Associates. The new owners took SASI's leading software, called ANSYS®, as their flagship product & designated ANSYS, Inc. as the new company name.
Ansys (Analysis System) is brilliant software used for very complex and crucial analysis which is playing vital roles in today's engineering. Basically it is engineering simulation software. This special software has no parallel application that could stand beside it.

BASIC CONCEPT FEM :

The basic idea in the finite element method is to search the solution of a complex problem by replacing it by a simpler one. Since the real problem is replaced by a simpler one in finding the solution, we will be able to search only an approximate solution rather than the right solution. The existing mathematical tools will not be sufficient to search the right solution (and sometimes, even an approximate solution) of most of the practical problems. Therefore, in the absence of any other convenient method to search even the approximate solution of a given problem, we have to prefer the finite element method. Moreover, in the FEM, it will be possible to improve or refine the approximate solution by spending more computational effort.

In the finite element method, the solution region is considered as built up of many small, interconnected sub regions called finite elements. As an example of how a finite element model might be used to show a complex geometrical shape, consider the milling machine structure shown in . Since it is very difficult to find the exact response (like stresses & displacements) of the machine under any specified cutting (loading) condition, this structure is approximated as composed of several pieces as shown in Figure 1.6(b) in the finite element method. In each piece or element, a convenient approximate solution is assumed & the conditions of overall equilibrium of the structure are derived.



NUMERICAL FORMULATION

A beam is a structural member used for bearing loads. It is typically used for resisting vertical loads, shear forces & bending moments.

TYPES OF BEAMS [18]:

Beams can be classified into many kinds based on 3 main criteria. They are as follows-

1. Based on geometry:

a. Straight beam – Beam with straight profile
b. Curved beam – Beam with curved profile
c. Tapered beam – Beam with tapered cross section
d. Based on the shape of cross section-

i. I-beam – Beam with 'I' cross section
ii. T-beam – Beam with 'T' cross section
iii. C-beam – Beam with 'C' cross section

2. Based on equilibrium conditions:

a. Statically determinate beam – For a statically determinate beam, equilibrium conditions alone can be used to solve reactions.
b. Statically indeterminate beam – For a statically indeterminate beam, equilibrium conditions are not enough to solve reactions. Additional deflections are needed to solve reactions.

3. Based on the type of support:

a. Simply supported beam
b. Cantilever beam
c. Overhanging beam
d. Continuous beam

II. METHODOLOGY

Ansys are founded on the concept of FINITE ELEMENT ANALYSIS. It is an approximate solution of Engineering Problems . There are three main approaches to constructing an approximate solution founded on the concept of FEA:

- **Direct Approach** This approach is used for relatively simple problems, and it usually serves as a means to explain the concept of FEA and its important steps.
- **Weighted Residuals** This is a versatile method, allowing the application of FEA to problems that's functional cannot be constructed. This approach directly utilizes the governing differential equations, such as those of heat transfer and fluid mechanics.
- **Vibrational Approach** This relies on the calculus of variations, which involves extremizing a functional. This functional corresponds to the potential energy in structural mechanics.

OBJECTIVE AND SCOPE OF WORK:

In this report, we will be formulating the bending equations of beams. The bending stress of various beams system will be found out at different variables of beam using ANSYS 15 and use Taguchi Method. The results will be compared further using analytical method of various beams. The objectives are following –

1. Considering different type of the beam* and draw the Bending Test Setup (Such as Fig. 3.1).
2. To Simulate the Bending Test Setup at Ansys Workbench 15.0
3. Find out The Deflection And Bending Stress Generated by Ansys Workbench 15.0
4. Calculate The Deflection and Bending Stress Using Mathematical Formulas.
5. Prepare A Report Generated by Ansys Workbench 15.0
6. Find out the impact of loads at different materials.
7. Plot the graph between Mathematical and Ansys Values.

MATERIAL PROPERTIES:

For each element type, there are a minimum number of required material properties. This number depends on the kind of analysis. The material nature may be:

- Linear or nonlinear.
- Isotropic, orthotropic, or anisotropic.
- Temperature dependent or independent.

TAGUCHI METHOD

Taguchi Method is a new engineering design optimization methodology that improves the quality of existing products and processes and simultaneously reduces their costs very rapidly, with minimum engineering resources and development man-hours. The Taguchi Method achieves this by making the product or process performance "insensitive" to

variations in factors such as materials, manufacturing equipment, workmanship and operating conditions.

Taguchi methods (Japanese) are statistical methods, or sometimes called robust design methods, developed by Genichi Taguchi to improve the quality of manufactured goods, and more recently also applied to engineering,[2]

Taguchi's use for manufacturing

Taguchi realized that the best opportunity to eliminate variation of the final product quality is during the design of a product and its manufacturing process. Consequently, he developed a strategy for quality engineering that can be used in both contexts. The process has three stages:

- System design
- Parameter (measure) design
- Tolerance design

III. CONCLUSION

The present work has been carried out to study the deflection of beam on applied load of 5000 N to 10000 N of simply supported beam with UDL, simply supported beam with point load and applied load of 500N to 1000N on cantilever beam. The calculation has been made theoretically as well as using ANSYS WORKBENCH 15 and TAGUCHI METHOD. Taguchi method applied on the result of deflection for validation of deflection result use for manufacturing technique. From the obtained result it can be concluded that out of Aluminum alloy 7075-T6, Gray cast Iron, Stainless steel 305 and Structural Steel 345w, has minimum deflection and maximum bending stress in all loading conditions.

A. Material Study:

I. We took the different materials Aluminium Alloy 7075-T6, stainless steel 305and Structural Steel 345w.
II. specified the material properties, applied meshing and boundary condition.

B. Mathematical Investigation:

According to beam and load type we applied the mathematical formula and then got the deflections and stresses.

C. Ansys Investigations:

We got very closer values of the deflections and stresses through the ansys workbench.

D. Suggestions for Future Study:

I. We can investigate to take any other cross-sections or different length of beams.

II. This can be extended to include coating of any materials, composite materials etc.

IV. REFERENCES

[1]. Singiresu S. Rao, the Finite Element Method in Engineering, 2011, Butterworth-Heinemann,USA

[2]. Rosa, Jorge Luiz; Robin, Alain; Silva, M. B.; Baldan, Carlos Alberto; Peres, Mauro Pedro Journal of Materials Processing Technology. 209

[3]. Kailash C. Bhosale"Investigates Analysis of bending strength of helical gear"(Maharashtra) India-Vol 2, No 4, 2011

[4]. Ratnadeepsinh M. Jadeja,Dipeshkumar M. Chauhan, Jignesh D. Lakhani"Investigates bending stress analysis of bevel gear" IJIRSET Vol. 2, Issue 7, July 2013

[5]. Ahmad O. Moaaz "Studies Finite element stress analysis of a heavy truck chassis plays an important role during design stages"IJAET Nov., 2014

[6]. AbulHasan"Studies finite element analysis and fatigue analysis of spur gear under random loading".

[7]. D.Bharadwaj"Investigates numerical study on stress analysis of curved beams"Volume 6, Issue

7, July (2015), pp. 21-27 Articles ID: 30120150607004

[8]. AnkitGautam"Investigates modal analysis of beam through analytically and FEM" YMCA New Delhi, 27th May 2016

[9]. Abdul RidahSaleh"Investigates about the experimental and Mathematical analysis for behavior of R.C" Vol.(21): 2013

[10]. Chaphalkar"Investigates about the modal analysis of cantilever beam structure using finite element analysis and experimental analysis"(AJER), 2015

[11]. AnkitGautam"Studies about the modal analysis of beam through analytically and FEM" 27 May 2016

[12]. Kailash C. Bhosale"Investigated that in helical gear the engagement between driver gear and driven gear teeth begins with point contact and gradually extends along the tooth surface"Volume 2, No 4, 2011

[13]. Ratnadeepsinh M. Jadeja "Investigate the analytical calculation and analysis in ANSYS of straight teeth bevel gear"Volume 2, 2013

[14]. P. Moulali"Acomparative study between composite leaf spring and steel leaf spring with respect to stress, deformation, life, and damage and safety factor"March 2014.

[15]. ShaikGulamAbulHasan"Research a modal analysis on spur gear under random loading & fatigue analysis" July, 2015

[16]. Sameer Chakravarty NC & B Subbaratnam"They are analysed for random even under loading & continuous dimension fatigue analysis & loading conditions for life gear has been obtained by the finite element ANSYS package" 2014

[17]. D. Bharadwaj, A. Purushotham"Research a numerical study on stress beams ANSYS software to determine these values for curved beams, cross section selected for curved beam is a rectangle"Volume 6, Issue 7 July (2015)

[18]. http://mechteacher.com/beam/#ixzz4aEQpmGuD (accessed 26 Feb. 2017)

[19]. ErdoganMadenci and Ibrahim Guven, 2015, the Finite Element Method and Applications in Engineering Using ANSYS, Second Edition, Springer, Yew York

Cite this article as :

Sh