SKNSITS_RTME - 2016 ISSN: 2321-8134

IJFEAT

INTERNATIONAL JOURNAL FOR ENGINEERING APPLICATIONS AND TECHNOLOGY

Finite Element Approach to Evaluate Strain Energy Absorbed by Specimen in Tensile Test

Rahul V Mahajan
Asst Professor
SKN Sinhgad Inst of Tech & Sci Lonavala
Maharashtra (India)
rvmahajan@live.in

Durvesh V Dandekar
Asst Professor
SKN Sinhgad Inst of Tech & Sci Lonavala
Maharashtra (India)
durvesh.dandekar@gmail.com

***Abstract*: The Finite Element Approach is extremely suitable in cases where the boundary conditions are complicated, the loading is abnormal and material discontinuity is present. This makes the use of FEA packages very popular for engineers and researchers. This paper demonstrates the use of ANSYS to calculate the strain energy absorbed in Specimen loaded in tension. The results are compared with Rayleigh-Ritz method and further they are validated by actual testing of specimen on UTM with data acquisition capability. This paper is mainly aimed to make the beginners understand the theory underlying FEA.**

***Index Terms*—FEA, Strain Energy, Rayleigh-Ritz method**

Introduction

The Finite Element method has gained its potential from the variety of problems it can solve. Further, the problems for which the analytical solutions are impossible e.g. boundary value problems involving material discontinuity, geometric discontinuity can be easily modeled with the use of Finite Element Methods. Engineering components often includes stress risers like notches, fillets, the stresses near these discontinuities can be easily found out using FEA softwares. Apart from this, use of FEA method for problem analysis yields much more practical solutions than those obtained from use of analytical one. Brahmbhatt Dipen et.al in their paper “Stress Concentration Factor Converts Into Stress Intensity Factor Using ANSYS” showed that at lower values of aspect ratio, stress intensity Factor must be taken into account and not the stress concentration factor [1]. Obviously, this gives researchers more practical way of dealing with problems. The use commercial FEA packages without sound knowledge of FEA is not advocated. Unless user has the background of FEA, he may produce worst results and may go with overconfidence [2]. In this method, the author has tried a simple problem of tensile testing of round specimen. The results of testing i.e. Youngs modulus E as obtained from test forms input to ANSYS. The results of Ansys are further compared with solution of the same problem using variational method i.e. Rayleigh-Ritz method. This will give the beginner a clear understanding of problem and will create base for him to use FEA for analysis that is much more complex. The approach used here in this paper is theoretical calculation of strain energy using Principle of stationary potential. The same is calculated using ANSYS software and these results are compared with actual testing results. One might argue that for such calculation weather to use engineering stress strain curve or true stress strain curve. We will use engineering stress strain curve, as our analysis is mainly limited to elastic region. For analysis of material in plastic region, we need to use true stress strain curve; also there is not much variation between engineering stress strain curve and true stress strain curve up to limit of proportionality [3][4]

I Material And Method

The test specimen is tested in simple tension test on UTM with data acquisition capability. The value of Young's modulus (E) is found to be 8.5 Gpa.

The reason for low Value of Young's modulus is that, the specimen had poor surface finish. Surface condition of specimen has direct impact on strength of material [5] The value of Young's modulus is taken as input data for ANSYS.

Now, the another specimen with same surface finish is loaded on UTM the details of test specimen were

1 Specimen diameter = 9.1 mm
2 Gauge length = 160 mm
3 Yield point load = 35.45 KN
4 Yield point elongation = 13.720 mm
5 Tensile strength = 704.426 Mpa
6 % Elongation = 29.69

Figure 1 Tensile test specimen

The test is carried on UTM the graph shown in figure 2 shows behavior of material under pure tensile load. From the graph obtained, the strain energy stored in material up to a loading of 20 KN can be found out as follows.

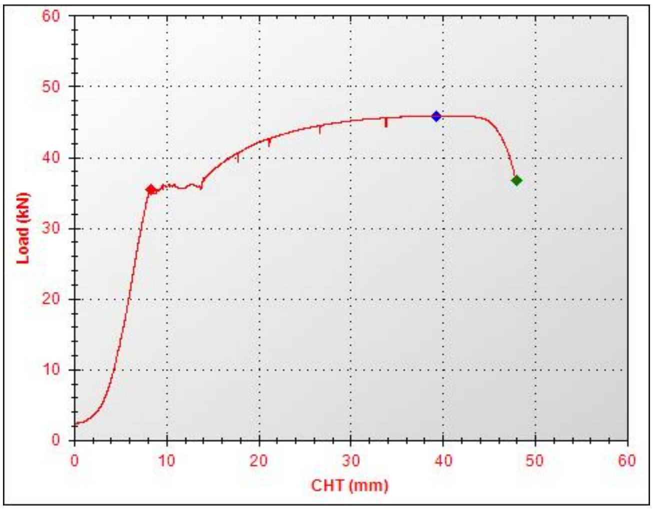

Figure 2 Load Vs Cross Head Travel (Deflection)

U = strain energy = ½ load x deflection= Area under load deflection curve = ½ x 20000 x 5.7 = **57 N-mm.**

II PROBLEM SOLVING USING VARIATIONAL METHOD

Now, the problem is solved by Raleigh -Ritz method. The problem is considered as 1D problem

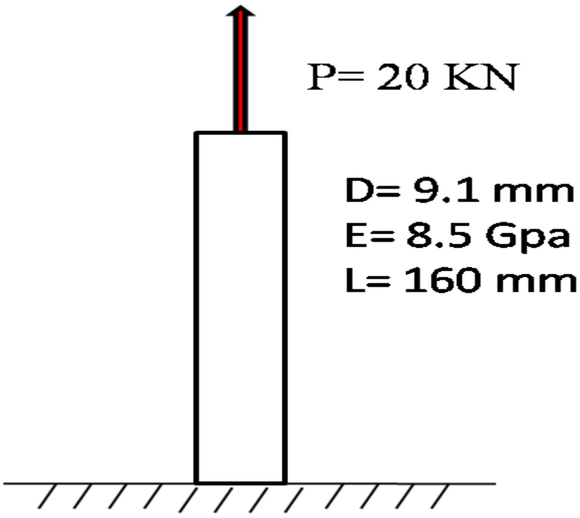

Figure 3 Modeling as 1D problem

here the unknown displacement field u is approximated to vary in quadratic manner as follows

$u = Cx + D x^2$

Where, C and D are unknown constants which satisfy boundary conditions i.e u=0 at x=0 where x is measured from fixed end in the direction of length of specimen.

Now the potential energy Π for specimen rod is given as

$$\Pi = \frac{1}{2}\, EA \int \varepsilon^2\, dx - P \times u_{(x=L)}$$

Where $\varepsilon = \frac{du}{dx}$ is strain in x direction.

Thus, $\varepsilon = \frac{du}{dx}$ = C +D x

Substituting the value of strain in above equation and solving we get,

$$\Pi = \frac{1}{2} EA \left[C^2 L + 2CDL^2 + \frac{4}{3} D^2 L^3 \right] - P[CL + DL^2]$$

Now, form the principal of minimum potential energy, $\frac{\partial \pi}{\partial C} = 0$; $\frac{\partial \pi}{\partial D} = 0$ solving these equations we get

$2C + 2LD = \frac{2P}{AE}$ -------(a)

$2C + \frac{8}{3} LD = \frac{2P}{AE}$ -------(b)

Observing these two equations, we can conclude that value of D must be zero. i.e the field variable must be linear for point load. (It can be quadratic for traction like forces) [5]

Solving above, we get **C= 0.036175 & D= 0**

Thus, strain energy U corresponding to a load of 20 KN is obtained as 57.877 **N-m.** The strain energy depicts the internal work done by resistance developed against the straining of material due to application of load.

III Modeling problem in ANSYS

Now, the same problem is modeled in ANSYS APDL software. The element used is link 180 and this problem can be modeled as 1D problem [6][7]. The one end of rod is fixed and other end is subjected to a point load of 20 KN. The results of ANSYS are shown in figure 4 and 5 below

Thus from ANSYS, The strain energy absorbed is as follows

U = Strain energy absorbed = **57.883 N-m**

Note that, we have taken only one element to model the problem in ANSYS. Hence, the elemental solution will give the total value of strain energy absorbed by the rod. The analysis can be performed by taking suitable no of elements for given element type (if permitted by software) in that case, the total strain energy is sum of strain energy of all the elements present in the problem.

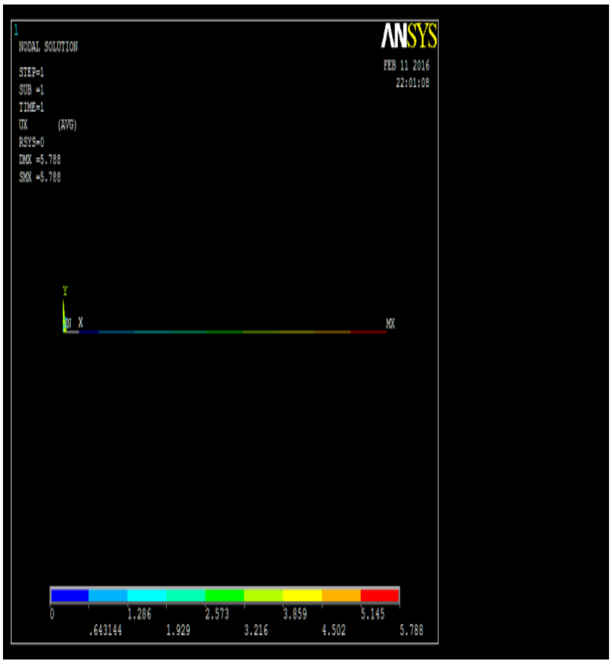

Figure 4 Deflection as obtained from ANSYS

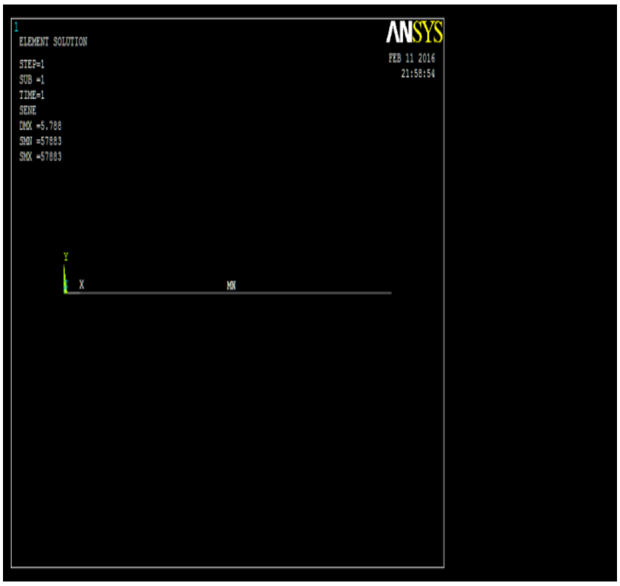

Figure 5 Strain energy as obtained from specimen

IV CONCLUSION

From the above discussion, we find that, the strain energy as obtained by Principle of stationary potential and ANSYS is same as that is measured from actual testing. Also for point loading (load P at free end and other end fixed,) the variation of field variable is linear. Below is summary of results of all three methods

Sr. no	ANSYS Result	Rayleigh-Ritz method	Tensile Testing
1	57.883 N-m	57.877 N-m	57 N-m

REFERENCES

[1] Brahmbhatt Dipen, Brahmbhatt Ketul and Patel Dipal, "Stress Concentration Factor Converts Into Stress Intensity FactorUsing ANSYS" 2015, 2(1): 46-49,EJAET, ISSN: 2394 - 658X pp 46-49

[2] S S Bhavikattil, "Finite Element Analysis, NewAge International Publisher"

[3] S L Kakani, Amit Kakani, "Material Science", New age International Publisher ISBN (13) : 978-81-224-2656-4 pp 228-238

[4] K Mahadevan, K balaveera Reddy," Design data handbook" third edition ISBN 81-239-0162-3

[5] P Seshu, "A text book of Finite Element Analysis." PHI, ISBN 978-81-203-2315-5 pp 76-77

[6] T. Stolarski, Y Nakasone, S Yoshimoto,"Engineering analysis with ansys software" Elsevier Butterworth-Heinemann ISBN 0 7506 6875 X

[7] www.ansys.belcan.com