International Journal of Science and Research (IJSR)
ISSN (Online): 2319-7064
Index Copernicus Value (2016): 79.57 | Impact Factor (2017): 7.296

Static Analysis of a Tapered Beam Using Excel and Ansys

Gajja Naga Prasanna

Abstract: ***Beams are slender members that are used for supporting transverse loading, and produces significant bending deformation measured as a transverse displacement and a rotation. Long horizontal members used in buildings and bridges and shafts supported in bearings are some examples of beams. Degrees of freedom at each node of a beam element are transverse displacement and rotation. A beam is a long prismatic bar either in 2D or in 3D space, designed to support external pressures and forces. The external forces may be perpendicular or inclined at an angle to the axis of the beam. Beams are normally considered to be horizontal and straight, whose one dimension is longer than the other dimensions. In this paper we present analysis of a tapered beam using finite element method concepts. Extracting the results using Excel and comparing the results using ANSYS software. The main objectives of this thesis are to calculate the displacement and stresses for the tapered beam using excel and comparing the results gained using ANSYS.***

Keywords: Static analysis, Tapered beam, Fem, Excel, ANSYS

1. Introduction

Structural analysis is used to access the behavior of engineering structures under the application of various loads. Commonly used structural analysis methods include analytics methods, experimental methods and numerical methods. Analytical methods provide accurate solutions with applications limited to simple geometries. Experimental methods are used to test prototypes or full scale models. However, they are costly and may not be feasible in certain cases. Among many numerical methods, finite element analysis is the most versatile and comprehensive numerical technique in the hands of engineers today.

The various steps involved in element analysis are:

1) Select suitable field variables and the elements
2) Discritise the continua.
3) Select interpolation functions
4) Find the element properties
5) Assemble element properties to get global properties
6) Impose the boundary conditions
7) Solve the system equations to the nodal unknown
8) Make the additional calculations to get the required values

2. Theoretical Study of Tapered Beam

The first step in the finite element method is to divide the structure into subdivisions or elements. Hence the structure is to be modelled with suitable finite elements. The number, size and arrangement of the elements are to be decided. Since the displacement solution of a complex structure under any specified load conditions cannot be predicted exactly, we assume some suitable solution within an element to approximate the unknown solution. The assumed solution must be simple from a computational point of view, but it should satisfy certain convergence requirements. In general the solution or the interpolation model is taken in the form of a polynomial.

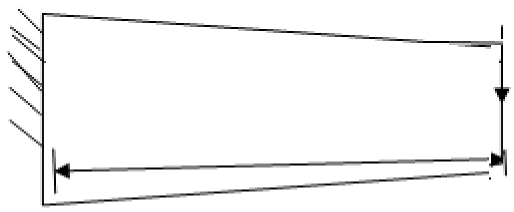
Developing stress/strain and strain/displacement relationship.

Derivation of stiffness matrix.

From the assumed displacement model, the stiffness matrix and load vector of the element are to be derived using either equilibrium conditions

1) Direct stiffness method
2) Work-energy method
3) Weighted-residual method

Since the structure is composed of several finite elements, the individual element stiffness matrices and load matrices and load vectors are to be assembled in a suitable manner and the overall equilibrium equations have to be formulated as F=K.X. Formulation of governing differential equation of tapered beam, A general Euler's Bernoulli beam is considered which is tapered linearly in horizontal. F



The width and depth are varying linearly given by

$h = h_1 + (h_0 - h_1)(x/l)$

$b = b_1 + (b_0 - b_1)(x/l)$

Similarly area and moment of inertia will be varying accordingly

$A(x) = (h_1 + (h_0 - h_1)(x/l))(b_1 + (b_0 - b1)(x/l))$

$I(x) = I/12\,[b_1 + (b_0 - b_1)(x/l)][h_1 + (h_0 - h_1)(x/l)^3]$

All the expressions for the beam area and moment of inertia at any cross-section are written after considering the variation along the length to be linear. Where ρ is the weight density, A is the area and together 'ρA/g' is the mass per unit length, E is the modulus of elasticity and I is the moment of inertia and L is the length of the beam.

$$K = E\begin{bmatrix} K_{11} & K_{12} & K_{13} & K_{14} \\ K_{21} & K_{22} & K_{23} & K_{24} \\ K_{31} & K_{32} & K_{33} & K_{34} \\ K_{41} & K_{42} & K_{43} & K_{44} \end{bmatrix}$$

Where

$K_{11} = 6(I_0+I_1)/IK_{22} = I_0+3I_1/I$

Licensed Under Creative Commons Attribution CC BY

$K_{12} = 2(I_0+2I_1)/I^2 = k_{21}K_{23} = -2(I_0+2I_1)/I^2 = K_{32}$
$K_{13} = -6(I_0+I_1)/I^3 = k_{31}K_{24} = I_0+I_1 /I = K_{42}$
$K_{41}=k_{14}= 2(2I_0+I_1)/I^2 K_{34} = 2(2I_0+I_1)/I^2 = K_{43}$
$K_{33} = 6(I_0+I_1)/I^3 K_{44} = 3I_0+I_1 /I$

I_0 and I_1 are calculated using these formulae given below

$I_0 = b_0 d_0^3 /12$
$I_1 = b_1 d_1^3 /12$

The above equation is called as the element stiffness matrix for tapered beam with rectangular cross sectioned area. Considering a tapered beam of length 1000mm with depths d_0=90mm and d_1=40mm and width $b=b_1=b_2$=20mm

			h0=	90	
stress=	591.719766	x3=	3.642E-05	f3=	0
		x4=	6.575E-05	f4=	0
		x5=	0.7493942	f5=	0
		x6=	0.006611	f6=	0
		x7=	2.4179508	f7=	0
		x8=	0.0096977	f8=	0
		x9=	4.520113	f9=	0
		x10=	0.0111356	f10=	0
		x11=	9.1881597	f11=	2000
		x12=	0.0119586	f12=	0

1=	2.103E-09
2=	-1.4E-09
3=	0
4=	0
5=	0
6=	0
7=	0
8=	0
9=	2000
10=	0

obj=	16.91512275

A load of 2KN is applied at the small end of the tapered beam. Taking E=2x10^5 N/mm^2

The stress of a tapered beam when a load of 2KN applied is 591.719766 N/mm^2

The obtained displacement value is 16.91512275.

Task II
Introduction to ANSYS:
ANSYS is a general purpose finite element analysis software developed supported and marketed by ANSYS, Inc., founded in 1970 by Dr. John A. Swanson. ANSYS Inc. is a global innovator of simulation software and technologies providing a powerful suite of engineering simulation tools which accelerates the product development process at less computational and financial expenditure. ANSYS simulation solutions are used by several fortune 500 companies to produce a wide range of products, including aircraft and automobile engineers, spacecraft, computer chips, buildings, office furniture and medical devices. It is a package for numerically solving a wide variety of mechanical problems. These problems include static/dynamic structural analysis.
In general, a finite element solution may be broken into the following three stages. This is a general guideline that can be used for setting up any finite element analysis

1) Pre-processing :defining the problem the major steps in pre-processing are given below
   - Define key points/lines/areas/volumes
   - Define element type ad material/geometric properties
   - Mesh lines/areas/volumes/as required the amount of detail required will depend on the dimensionality of the analysis (i.e. 1d, 2d, axis symmetric, 3d).

Solution: assigning loads, constraints and solving; here we specify the loads and finally solve the resulting set of equations.

Post processing: further processing and viewing of the results; in this stage one may wish to see: (a) list of nodal displacements.
(b) Element forces and moments
(c) Deflection plots
(d) Stress contour diagrams

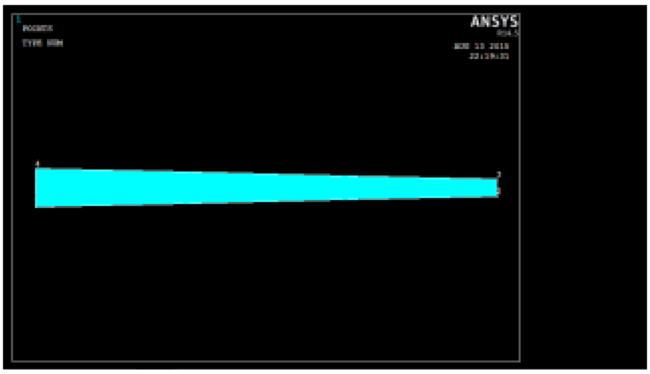
Tasks in Static Analysis:

The procedure for static analysis involves three main tasks:

1) Build the model
2) Apply loads and obtain solution
3) Review the results

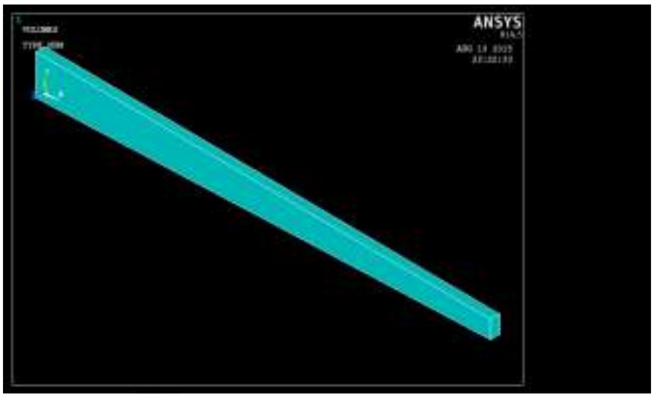
Procedure to design and analysis in ANSYS
Open preferences
Pick structural and h-method>ok
Define the type of element
Pre-processor > add/edit/delete…
For this problem we use solid brick 8node 185> ok
Define element material properties
Material properties > structural > linear > elastic > isotropic > EX 2E5 >prxy> 0.3. [Young's modulus 200000 N/mm^2, Poisson ratio= 0.3]
Modelling: Keypoints > in active cs > keypoint:

Areas > arbitrary > through key points
Select key points 1, 2, 3, 4 and ok

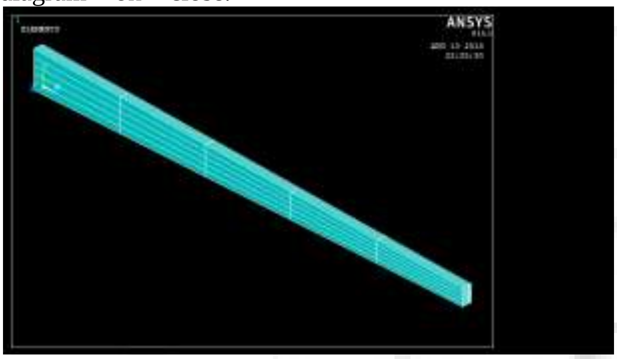


Operate extrude >areas > by xyz offset
Select the diagram > ok at DZ > 20 > ok [extrusion 20 mm]

Licensed Under Creative Commons Attribution CC BY

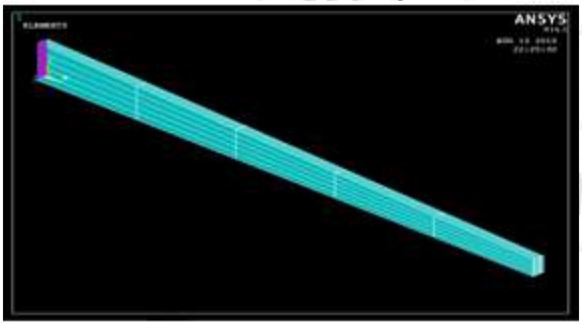


Defining mesh:
Meshing > size controls > manual size > global > size > no. of element divisions > 5 > ok
Mesh > volumes > mapped > 4 to 6 sided > select the diagram > ok > close.

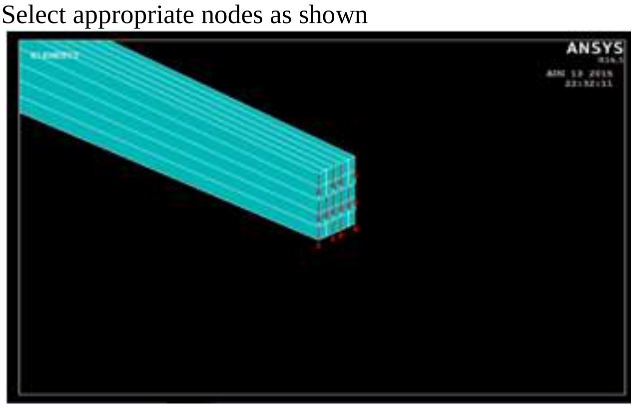


Defining the loads:
Loads > analysis type > new analysis > static analysis > ok
Define loads> apply > structural > displacement > on areas > ok > all Dof ok.

Select the area to be fixed as shown in fig

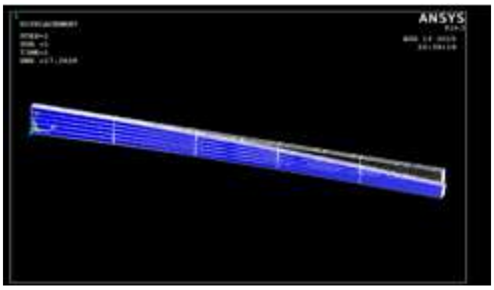


Force/moment > on nodes
Select appropriate nodes as shown



After selecting the nodes as shown apply load 2000N in FY direction downward i.e. -2000 for down ward position.
Solve the system:
Solve > current load step > yes > close.
Close the solution menu

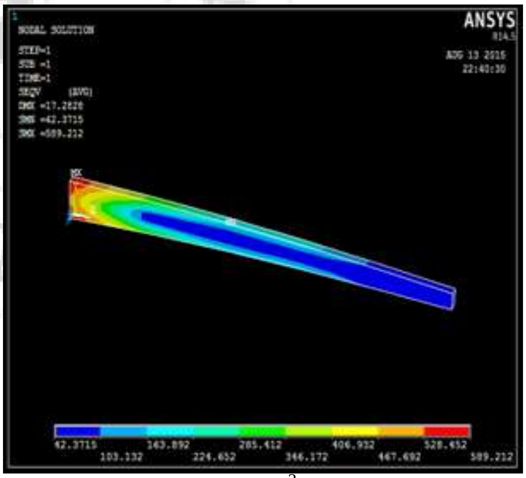
Viewing the results:
General post processor > plot results > deformed shape >deformed+undeformed> ok



Nodal Solution Per Node

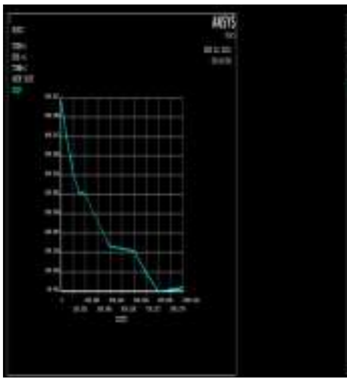
NODE	S_1	S_2	S_3	SINT	SEQV
1	178.21	-34.940	-487.60	665.81	588.92
2	29.535	7.0271	-78.730	108.26	98.950
3	28.426	22.750	-315.19	343.61	340.81
4	7.7065	0.40962E-02	-201.64	209.35	205.60
5	16.971	13.441	-179.33	196.30	194.56

NODE	UX	UY	UZ	USUM
1	0.0000	0.0000	0.0000	0.0000
2	-0.57367	-17.272	-0.13709E-02	17.282
3	-0.30954	-0.78086	-0.69964E-02	0.84000
4	-0.53558	-3.0736	-0.77506E-03	3.1199
5	-0.65455	-6.7938	-0.39164E-02	6.8253

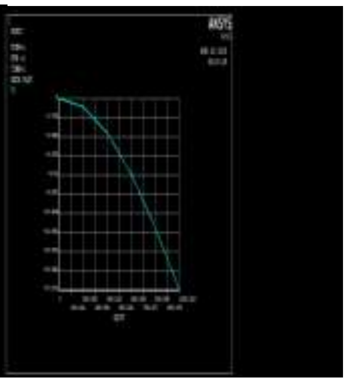


Maximum stress: 589.212 N/mm^2
Minimum stress: 42.3715 N/mm^2; deflection: 17.282

Licensed Under Creative Commons Attribution CC BY



Stress v/s distance graph



displacement v/s distance

3. Conclusions

Using EXCEL the calculated value of displacement of a tapered beam when 2kN load is applied on it is 16.91512275.

Using EXCEL the calculated value of stress of a tapered when 2kN load is applied on it is 591.719766 N/mm^2.

Displacement value obtained using ANSYS is 17.2828.

Minimum stress obtained for a tapered beam using ANSYS are SMN=42.3715 N/mm^2.

Maximum stress obtained for a tapered beam using ANSYS are SMX=589.212 N/mm^2.

References

[1] J.N. REDDY - An introduction to the Finite element method third edition chapter-5 analysis of beams and trusses.
[2] Introduction to finite elements in engineering third edition; Tirupathi R. chandrupatla and Ashok D. Belegundu
[3] Finite element analysis using ANSYS 11.0; Paletisrinivas, Krishna chaitanyasambana, rajeshkumardatti.
[4] Finite element method in engineering fourth edition; Singiresu S. Rao

www.ijsr.net IJSR ISSN (Online): 2319-7064

Licensed Under Creative Commons Attribution CC BY