International Journal of Innovative Technology and Exploring Engineering (IJITEE)
ISSN: 2278-3075, Volume-8 Issue-8, June 2019

Optimum Finite Element for the 1-D Geometrical Structures

A. Eswara Kumar, B.Kiran Kumar, B.V.Dhermendra, P.Phani Prasanthi

Abstract: ***Generally, in finite element method (CAE software like ANSYS) to solve the bar and truss problems two-node link element will be used. In 3-D link element, each node contains three DOF (Translations in x,y,z directions). To solve the beam structures, two-node beam element that has 6 DOF (3 Translations + 3 Rotations) will be used. In both theory and finite element CAE packages, it is creating slight confusion and trouble to use different elements for different structures as stated above. In the present work, an attempt was made to solve the bar and truss problems with beam element, as beam element already contains the DOF of link element. Finite element method simulation software was used to perform the analysis. Link 180 and beam 188 elements were used from the ANSYS library. The possibility of using the beam element for both bar and truss problems was verified for three different structures viz. stepped bar, plane truss and space truss in both static structural and modal analysis cases. From the results, it was observed that link element can be replaced with beam element conveniently for static structural and modal analysis cases with few assumptions.***

Index Terms**:** ***Bar, Truss, Link 180, Beam 188, Static Structural, Modal analysis;***

I. INTRODUCTION

Finite element method being a powerful analysis tool, is used to solve the practical problems [1-9]. Jiaxin solved statically determinate truss problem with finite element theory by using MATLAB [5]. Finite element method/ANSYS will contain different number of elements like link element, truss element, beam element, triangular, quadrilateral etc. Based on the type of the problem, different elements are used. Bars are the structural members subjected to axial loads only. They expand or compress due to external loads in the axial direction when treated as 1-dimensional elements. Truss members are connected by frictionless pin joints and are subjected to loads only at the joints. Generally, one end of the truss structure is supported by the hinged support and other end by the roller support. A plane truss structure can deform in a plane viz. XY plane, with truss members deforming axially. A space truss structure can deform in the space viz. truss members deforming in X or Y or Z direction. To solve these three structures, a two-node line element need to be used. In the finite element method simulation software like Ansys, link 180 will be used to solve these types of the problems. This element will contain three DOF per each node (3 Translations). Beam is a structural member generally

Revised Manuscript Received on June 7, 2019

A.Eswara Kumar, Department of Mechanical Engineering, Koneru lakshmaiah Education Foundation, Green Fields, Vaddeswaram,Guntur,

B.Kiran Kumar, Department of Mechanical Engineering, Koneru lakshmaiah Education Foundation, Green Fields, Vaddeswaram,Guntur,

B.V Dharmendra Department of Mechanical Engineering, Koneru lakshmaiah Education Foundation, Green Fields, Vaddeswaram,Guntur,

P.Phani Prasanthi, Department of Mechanical Engineering, PVPSIT, kanuru, Vijayawada, A.P

II. MODELLING

Stepped bar, plane truss and space truss were modelled in the Ansys software as shown in the figures 1, 2 & 3. Static structural and modal analyses were performed with link 180 and beam 188 elements.

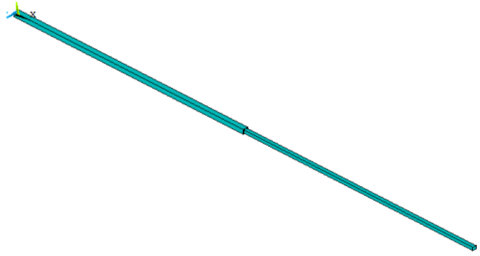


Figure 1: Stepped bar

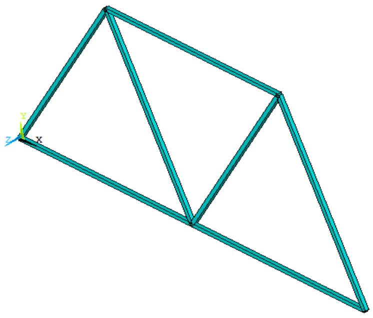


Figure 2: Truss Structure

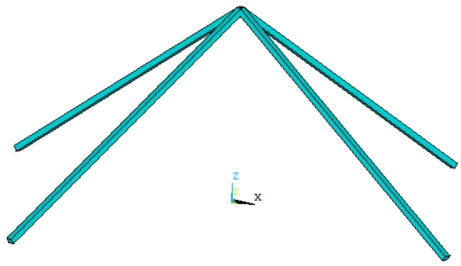


Fig 3: Space Truss

Link 180 is a two-node element with three DOF (all Translations) at each node, which is generally used to solve the bar and truss problems. The shape of the element is shown in the figure 4. Beam 188 is a two-node element with 6-DOF (3 translations + 3 rotations) at each node and is used to solve

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication

the beam and frame problems. The shape of the Beam 188 element is shown in the below figure 5.

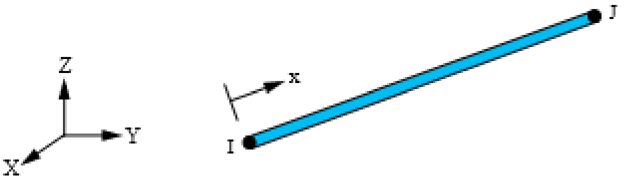


Fig 4: Shape of Link 180 Element

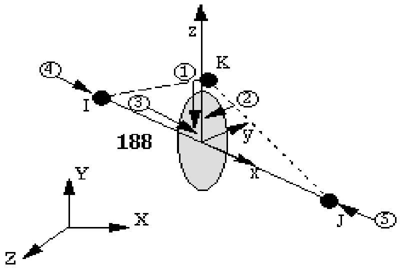


Fig 5: Shape of the Beam 188 Element

In the present study, all the bar, truss structures were considered as steel material with Young's modulus (E) as $2e^5$N/MM^2, Poisson's ratio (μ) as 0.3 and Density (ρ) as 7850kg/m^3

III. STATIC STRUCTURAL & MODAL ANALYSIS

In case of stepped bar, while using link 180 element, at end node all DOF and at all remaining nodes translation in Y and Z direction were constrained. With beam 188 element, all six DOF were fixed at one end and at all remaining nodes except translation in X direction remaining 5 DOF were fixed. Axial loads of 200KN and -300 KN were applied as shown in fig. 6.

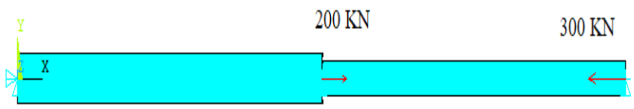


Fig. 6: Loads & Boundary Conditions for Stepped Bar

In case of plane truss, while using link 180 element, at end node of the truss structure, all DOF are constrained to represent hinged support and at the last node at the other end, one DOF regarding translation in Y direction is constrained to represent roller support and at all other nodes, translation in Z-Direction is constrained. With Beam 188 element, at all nodes rotations were constrained and the remaining translations at all nodes were constrained similarly to that of link 180 element. The loads of magnitude 200kN and 300kN are applied as shown in fig. 7.

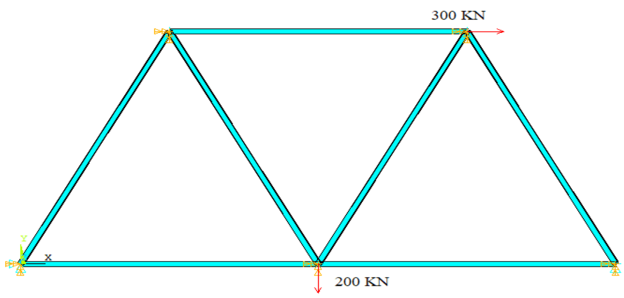


Fig:7 Loads & Boundary Conditions for Plane truss

In case of Space truss, all three nodes that are on the ground are constrained in all DOF in either cases while using link 180 or beam 188 element. Top node is constrained in all rotations with beam element and left unconstrained when using link 180. A load of magnitude 100kN is applied as shown in fig. 8.

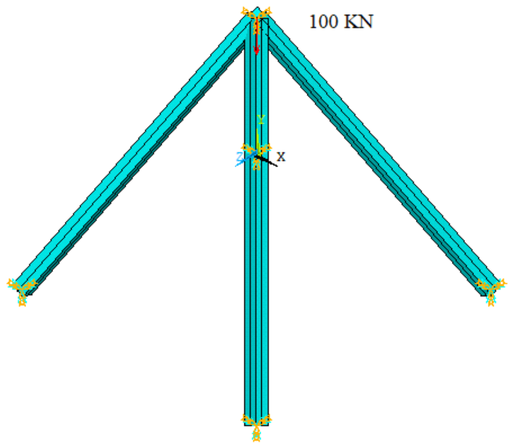


Fig 8: Loads & Boundary conditions for Space truss

Meshing, a process of converting geometrical entities into finite element entities, is the next step. A proper mesh is necessary for bar and truss problems when beam element is used. Each truss member and bar should be meshed with only beam element viz. only one division per element.
All the above three cases are analysed for deflection, elemental axial stress by performing static structural analysis. On all the above three models, modelling analysis is also performed to identify the natural frequencies.

IV. RESULTS & DISCUSSION

A. Deformation

The resulted deformation contour of the stepped bar with link and beam elements as shown in the fig.9 for Stepped bar, Fig.s 11 and 12 for plane truss, Fig.s 13 and 14 for space truss. The comparison of values of deformation is shown in the table 1 for all 3 cases considered. It was observed that both deformed shapes are same. The magnitudes of the deformations are very much similar, with a small deviation of negligible level. The slightest variation identified in the results are may be due to the element formulation for the both elements.

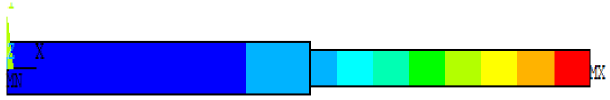


Fig 9: Deformation contour of Stepped bar (for Link Elements)

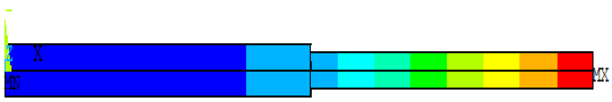


Fig 10: Deformation contour of Stepped bar (for Beam Elements)

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication

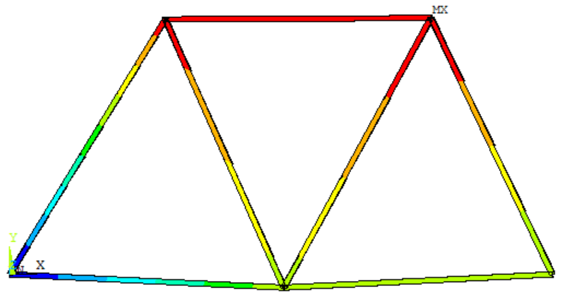


Fig 11: Deformation contour with link element

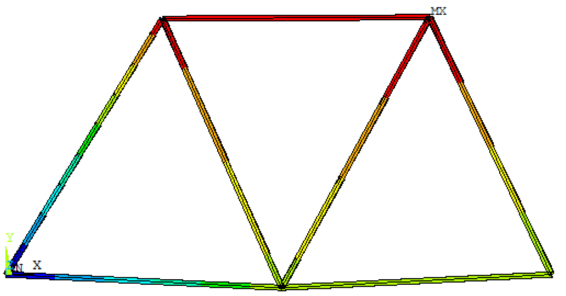


Fig 12: Deformation contour with beam element

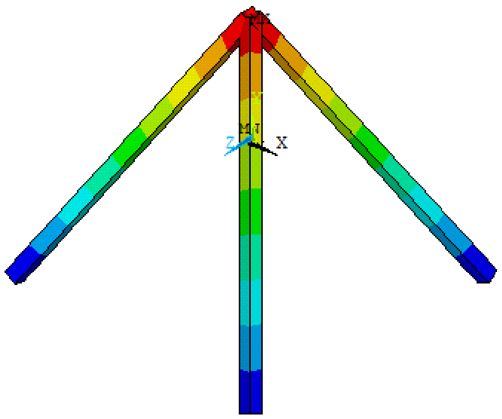


Fig 13: Deformation contour with link 180

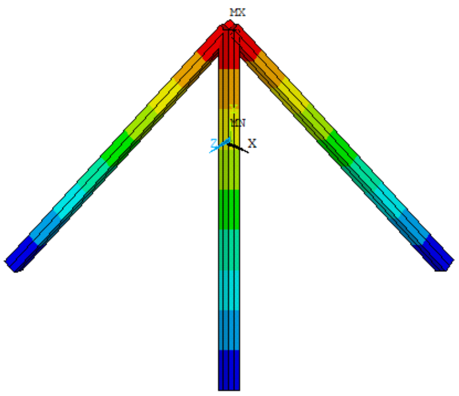


Fig 14: Deformation contour with Beam 188

Table 1: Deformation Values with link and beam elements

Structural Member	(Link 180)	Deformation (Beam 188)	% Variation
Stepped bar	3.56122mm	3.56122 mm	0
Plane Truss	17.8481 mm	17.7389 mm	0.61
Space Truss	0.931e-3mm	0.915e-3 mm	0.70

Table: 2 Reaction forces for the structures with link and beam elements

Structural Member	Link 180			Beam 188		
	Fx	Fy	Fz	Fx	Fy	Fz
Stepped Bar	-6122.45 =Max -1000= Min	0	0	0.1e6 @ node1	0	0
Plane Truss	-0.3e6	-50000 @Node 1 0.25e6 @Node 5	0	-0.3e6	-48809 @Node 1 0.24881e6 @Node 5	0
Space Truss	12.5	25	12.3	12.3	24.884	12.3

From the table 2, it can be observed that in case of stepped bar, and even in cases of plane truss and space truss, the magnitudes of reaction forces are very much similar with slight variation, which was due to constraining of rotational DOF in the beam element.

B. Element stresses

In case of stepped bar and plane and space trusses, the resulted contours of the stresses are shown in fig.s 15 to 20, and a comparison of magnitudes of axial elemental stresses are shown in table 3 for both link and beam element.
From these contours and values, it was observed that both the link and beam elements resulted in the same deformation, reactions and axial elemental stress values for 1-D bar type of problems.

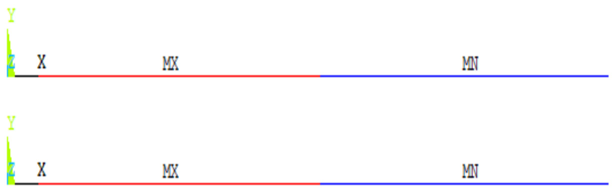


Fig: 15 & 16 Axial Stress in the Stepped Bar (Link & Beam Elements)

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication

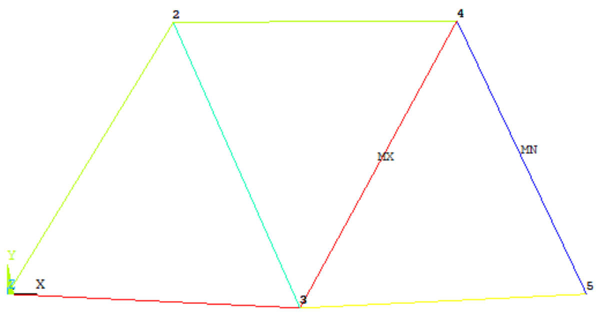


Fig:17 Stress in plane truss (link 180)

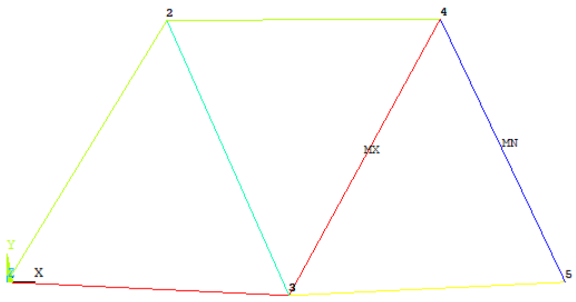


Fig: 18 Stress in plane truss (Beam 188)

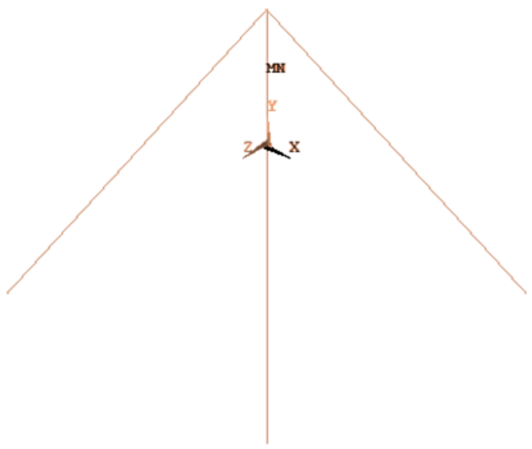


Fig:19 Axial Stress with link 180

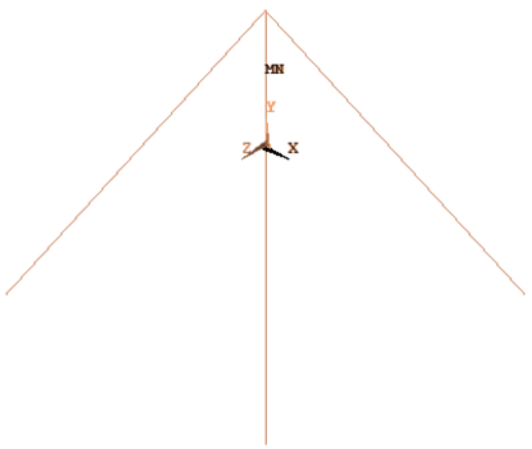


Fig:20 Axial Stress with Beam 180

Table:3 Comparison of Axial Stresses

Structural Members	Link 180 (Max, Min), MPa	Beam 188 (Max, Min)
Stepped Bar	-6122.45, -1000 N	-6122.45, -1000 N
Plane Truss	2795.08, -2795.08	2775.02, -2774.85
Space Truss	-1.2196	-1.2196

C. Modal Analysis

Modal analysis is generally used to perform to identify natural frequencies of the structure. These frequencies will depend on the elastic properties of the structure. In this study, first two natural frequencies are retrieved for the stepped bar, and first six natural frequencies are retrieved in case of plane truss and space truss, by using link 180 and beam 188 elements. In case of stepped bar, as the entire structure contains only three DOF, only two natural frequencies can be identified. Further, all the nodes in all 3 cases are constrained similarly as explained in static analysis while using either link 180 element or beam 188 element. No loading need to done in this case.

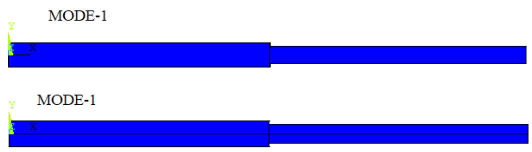


Fig :21 Mode-1 (Link 180 at Top & Beam 188 at Bottom)

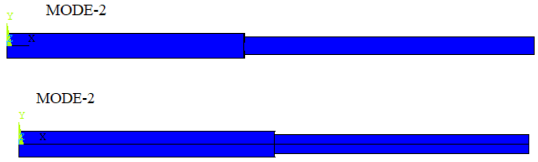
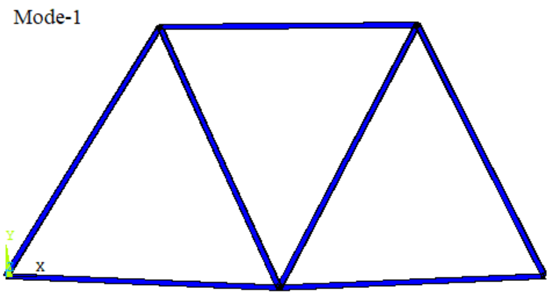
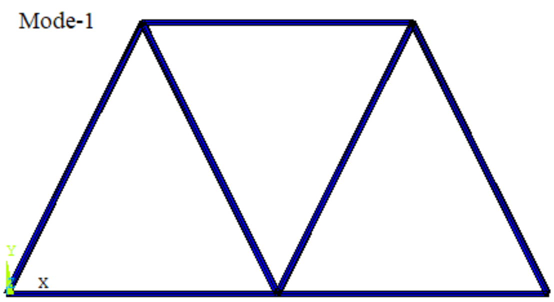


Fig :22 Mode-2 (Link 180 at Top & Beam 188 at Bottom)

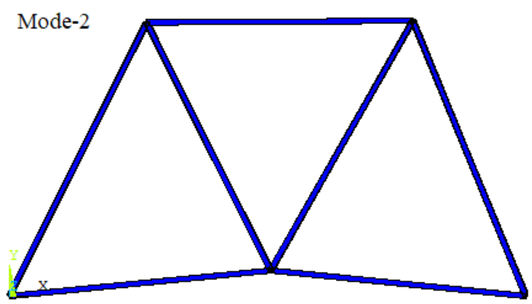


Mode-1 of Bar Element

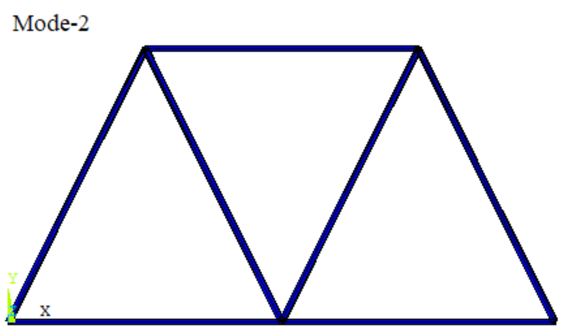
Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication



Mode-1 of Beam Element

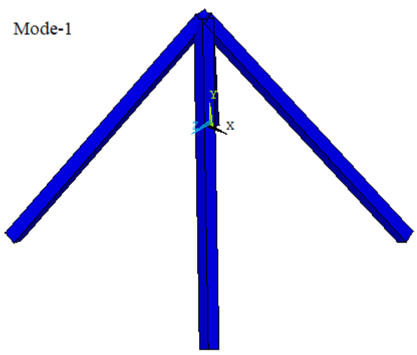


Mode-2 of Bar Element

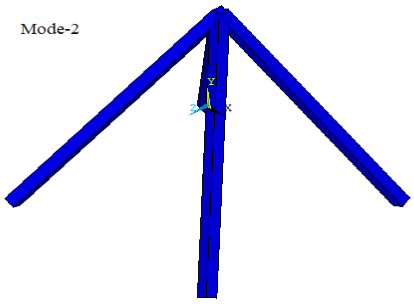


Mode-2 of Beam Element

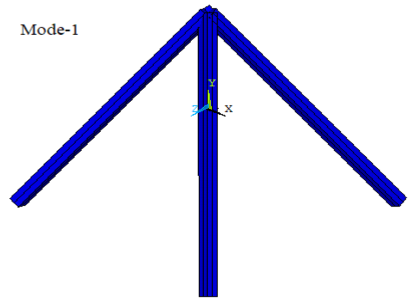
Fig :23 First Two Mode shapes of the truss structure with link 180 and beam 188 elements



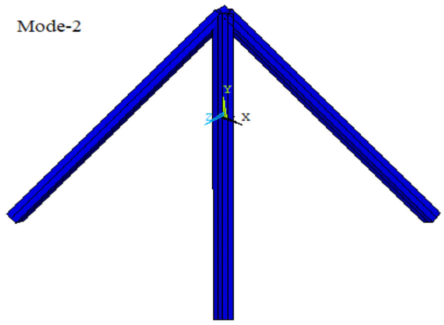
Mode-1 of Bar Element



Mode-1 of Beam Element



Mode-2 of Bar Element



Mode-2 of Beam Element

Fig :24 First Three Mode shapes of the space truss structure with link 180 (left) and beam 188 (right) elements

Table:4 Comparison of natural frequencies of Stepped bar

	Link 180	Beam 188
Mode-1	253.34	253.34
Mode-2	653.53	653.53

Table:5 Comparison of natural frequencies of Plane Truss

	Link 180	Beam 188
Mode-1	17.704	17.749
Mode-2	24.218	24.300
Mode-3	46.003	46.102
Mode-4	59.383	59.497
Mode-5	73.686	73.737
Mode-6	84.518	84.845

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication

Table:6 Comparison of natural frequencies of Space Truss

	Link 180	Beam 188
Mode-1	146.37	149.72
Mode-2	146.37	149.72
Mode-3	293.34	293.36

From the fig.s 21,22 and values of natural frequencies in table 4, it was observed that for both the elements the natural frequencies of mode-1 and mode-2 were same. It clearly shows that beam element can be replaced with link element for bar type of problems. From the fig.s 23,24 and values of natural frequencies in table 5 & 6, in case of Plane and space trusses, it was observed that both link and beam elements are resulting in the same results. The mode shapes are also same. It shows that beam 180 can be replaced with link 180 element for modal analysis also.

V. CONCLUSIONS

A compatibility study of using the beam element for both bar and truss problems in both static and modal analysis cases by using finite element simulation software ANSYS was done. From the results, it is recommended that the beam element can be used conveniently to solve the bar and truss problems for static and modal analysis with few assumptions viz. truss and bar members should be meshed with single element (divisions) to use with beam element and by constraining all rotational DOFs while using Beam element in place of bar element.

REFERENCES

1. Liu. X, "The finite element analysis of truss structure based on ANSYS", Advanced Materials Research, Vol.594-597, PP:2939-2944
2. Pramod Chaphalkar et al. "Introducing Finite Element Analysis In the first course of statics and solid mechanics", ASEE Annual Conference and Exposition, Conference Proceedings, 2007, 11p
3. Robert G.Ryan et al., "Development of Engineering Case Studies for Integrating Finite Element Analysis into a Mechanical Engineering Curriculum", ASEE Annual Conference and Exposition, Conference Proceedings, 2005, Pages 4481-4491.
4. C.J. Lissenden et al. "Applications of Finite Element Analysis for Undergraduates", ASEE Annual Conference and Exposition, Conference Proceedings, 2002, Pages 7701-7707
5. Jiaxin Zhao, "Teaching Finite Element Analysis as a Solution Method for Truss Problems in Statics", ASEE Annual Conference Proceedings, 2004, Pages 13567-13577.
6. William Edward Howard et al., "Finite Element Analysis in a Mechanics Course Sequence", ASEE Annual Conference Proceedings, 2001, Pages 5047-5055
7. Martin Pike, "Introducing Finite Element Analysis in Statics", ASEE Annual Conference Proceedings, 2001, Pages 6481-6484
8. Mamtimin Geni et al. "Modern numerical simulation methods and its practical applications in engineering", Gongcheng Lixue/Engineering Mechanics, Volume 31, Issue 4, April 2014, Pages 11-18.
9. Eswara Kumar A. et al. "Dynamic analysis of flex seal of solid rocket motor nozzle", Materials Today: Proceedings, Volume 4, Issue 2, 2017, Pages 1590-1597.
10. Karteek navuri, Eswara Kumar A,et al. "Random vibration analysis of mechanical hardware of flight data recorder", ARPN Journal of Engineering and Applied Sciences, Vol. 11, No. 18, Sep-2016.
11. Eswara Kumar A, Shahid Afridi,et al. "Finite element analysis of laminated hybrid composite pressure vessels", International Journal of Civil Engineering and Technology (IJCIET), Vol.8, Issue 4, April 2017, PP:916-934

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication