ISSN PRINT 2319 1775 Online 2320 7876

Research paper

 UGC CARE Listed (Group -I) Journal Volume 8, Issue 1, 2019

Unified Element Approach: Exploring Beam Elements for Bar and Truss Problems in Finite Element Analysis

D V A Rama Sastry

Dept. of Mech. Engg., Koneru Lakshmaiah Education Foundation, Vaddeswaram, Guntur, Andhra Pradesh, India

Abstract:

The conventional approach in finite element analysis involves utilizing a two-node link element to solve bar and truss problems within CAE software like ANSYS. These 3-D link elements, each comprising three Degrees of Freedom (DOF) – translations in the x, y, and z directions – present a standard methodology. Conversely, addressing beam structures involves employing a two-node beam element that encompasses six DOF (3 Translations + 3 Rotations). However, employing different elements for distinct structures (as noted above) often leads to confusion and challenges in both theoretical understanding and practical implementation within finite element CAE packages. This study endeavors to address this challenge by exploring the use of beam elements to solve bar and truss problems, leveraging the inherent DOF of beam elements. Through the utilization of finite element simulation software, this research delves into employing ANSYS library elements: Link 180 and Beam 188. The feasibility of utilizing beam elements for both bar and truss problems was scrutinized across three different structures: stepped bar, plane truss, and space truss. Static structural and modal analysis cases were considered for assessment. Analysis of the obtained results revealed that, with certain assumptions, the beam element could effectively substitute the link element in static structural and modal analysis scenarios. This observation suggests the potential for a convenient replacement of link elements with beam elements, offering a more unified approach in finite element analysis for bar and truss problems.

Keywords: Bar, Truss, Link 180, Beam 188, Static Structural, Modal analysis;

Introduction:

The finite element method, a robust analytical tool, is extensively employed to tackle practical problems [1-9]. Jiaxin utilized finite element theory via MATLAB to solve statically determinate truss problems [5]. Within the Finite Element Method (FEM) or ANSYS, diverse

ISSN PRINT 2319 1775 Online 2320 7876

Research paper UGC CARE Listed (Group -I) Journal Volume 8, Issue 1, 2019

elements like link, truss, beam, triangular, and quadrilateral are available, each suited for specific problem types.

Bars, acting as structural members under axial loads only, undergo axial expansion or compression due to external forces when treated as 1-dimensional elements. Truss members, interconnected by frictionless pin joints, solely bear loads at these joints. Typically, a truss structure is supported by a hinged end and a roller-supported end. A plane truss deforms within the XY plane, with its members deforming axially, while a space truss can deform in the X, Y, or Z direction, necessitating the use of a two-node line element for solving these structures.

In software like Ansys, the Link 180 element is commonly employed, featuring three Degrees of Freedom (DOF) per node (3 Translations) to solve such problems. Conversely, beams, subject to transverse loads, are addressed using two-node beam elements within the finite element method. In Ansys, the Beam 188 element, with six DOF per node (3 Translations + 3 Rotations), is employed to solve beam-related problems.

The objective of this study is to ascertain the feasibility of utilizing beam elements for both bar and truss problems in static and modal analysis scenarios, employing the finite element simulation software ANSYS.

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

ISSN PRINT 2319 1775 Online 2320 7876

 UGC CARE Listed (Group -I) Journal Volume 8, Issue 1, 2019

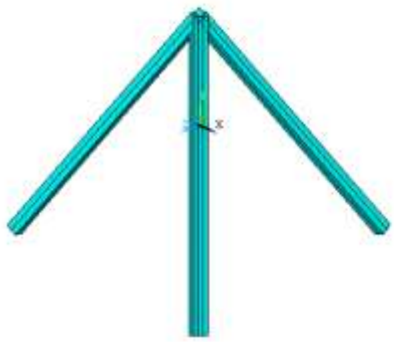
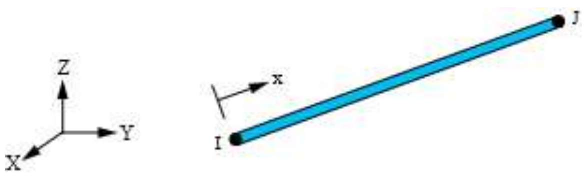


Fig 3: Space Truss

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



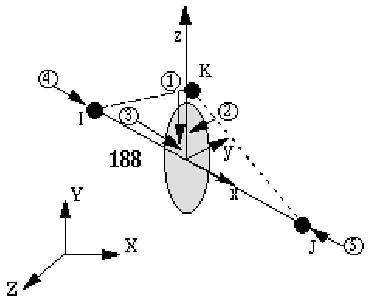


Fig 4: Shape of Link 180 Element

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 $7850 kg/m^3$

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.

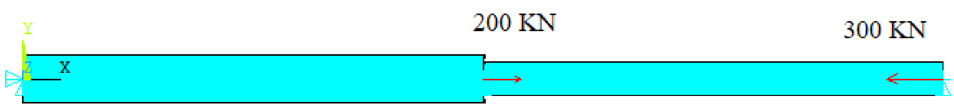


Fig. 6: Loads & Boundary Conditions for Stepped Bar

ISSN PRINT 2319 1775 Online 2320 7876

Research paper

 UGC CARE Listed (Group -I) Journal Volume 8, Issue 1, 2019

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.

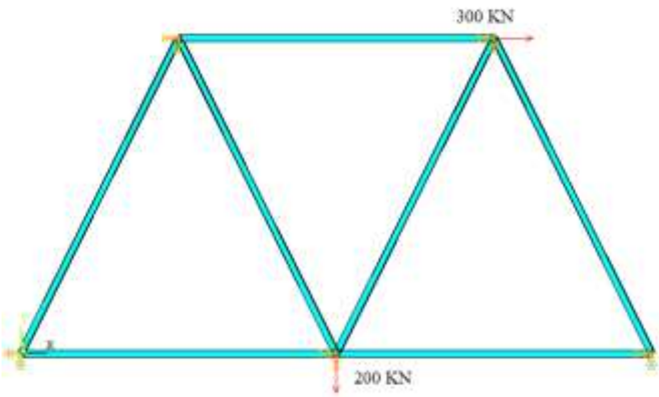


Fig:7 Loads & Boundary Conditions for Plane truss

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.

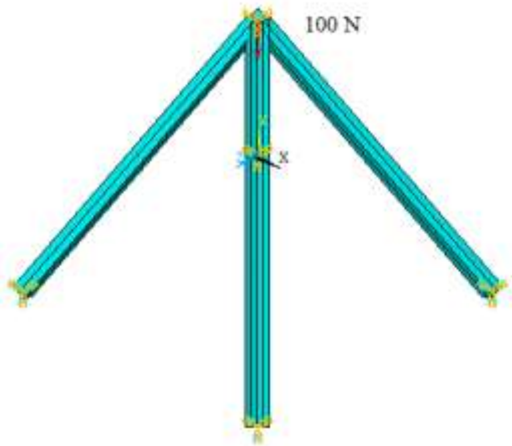


Fig 8: Loads & Boundary conditions for Space truss

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

ISSN PRINT 2319 1775 Online 2320 7876

Research paper UGC CARE Listed (Group -I) Journal Volume 8, Issue 1, 2019

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.

Results & Discussion:

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 comparision 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.

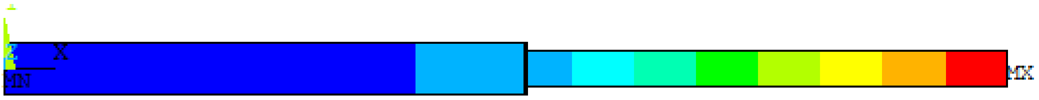


Fig 9: Deformation contour of Stepped bar (for Link Elements)

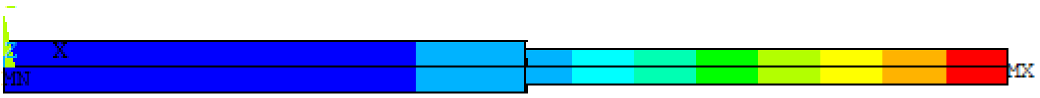
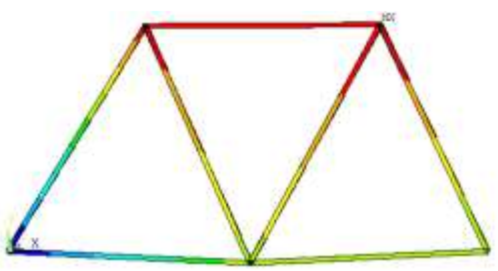


Fig 10: Deformation contour of Stepped bar (for Beam Elements)



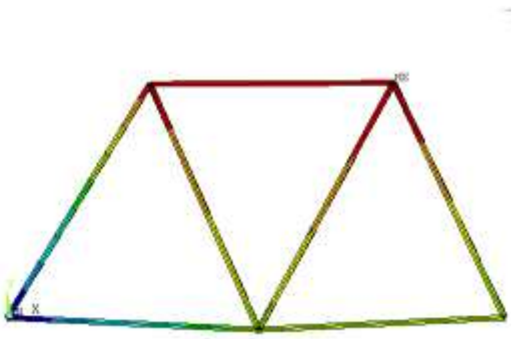


Fig 11: Deformation contour with link element   Fig 12: Deformation contour with beam element

ISSN PRINT 2319 1775 Online 2320 7876

Research paper

 UGC CARE Listed (Group -I) Journal Volume 8, Issue 1, 2019

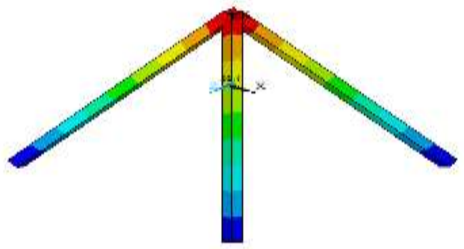


Fig 13: Deformation contour with link 180

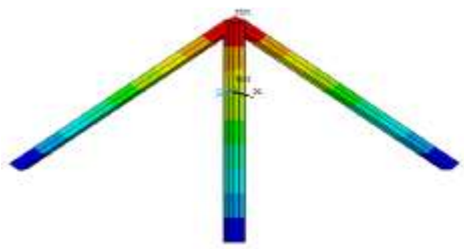


Fig 14: Deformation contour with Beam 188

Table 1: Total Deformation (MM) Values with link and beam elements

	Link 180	Beam 188	% Variation
Stepped bar	3.56122	3.56122	0
Plane Truss	17.8481	17.7389	0.61
Space Truss	0.931e-3	0.915e-3	1.71

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

	Link 180			**Beam 188**		
	F_x (N)	F_y (N)	F_z (N)	F_x	F_y	F_z
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	12.343	12.343	24.884	12.343

ISSN PRINT 2319 1775 Online 2320 7876

 UGC CARE Listed (Group -I) Journal Volume 8, Issue 1, 2019

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.

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 comparision 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.

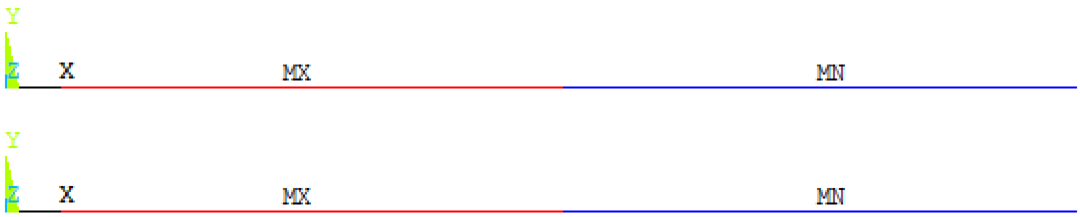
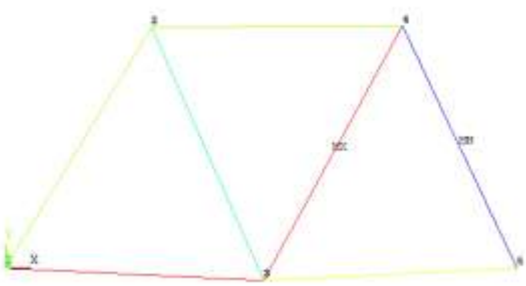


Fig: 15 & 16 Axial Stress in the Stepped Bar (Link & Beam Elements)



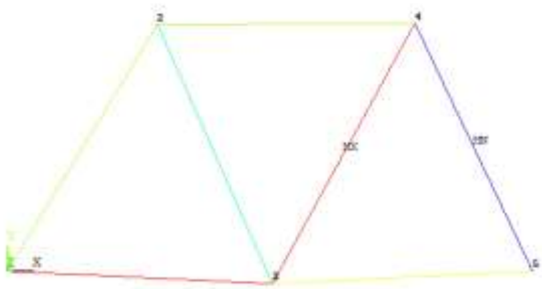


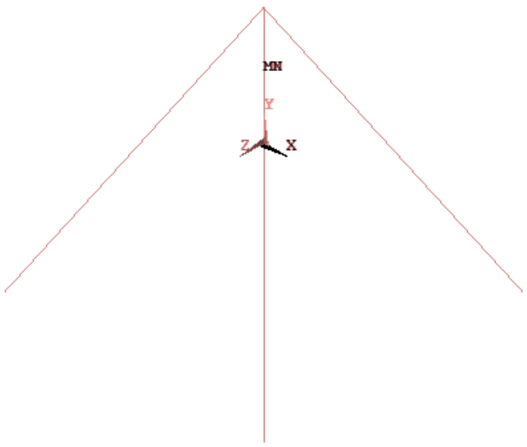
Fig:17 Stress in plane truss (link 180)

Fig: 18 Stress in plane truss (Beam 188)

ISSN PRINT 2319 1775 Online 2320 7876

Research paper

 UGC CARE Listed (Group -I) Journal Volume 8, Issue 1, 2019



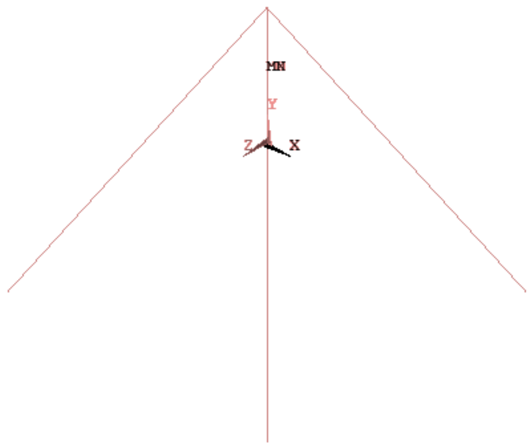


Fig:19 Axial Stress with link 180   Fig:20 Axial Stress with Beam 180

Table:3 Comparison of Axial Stresses (MPa)

	Link 180 (Max, Min)	**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

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.

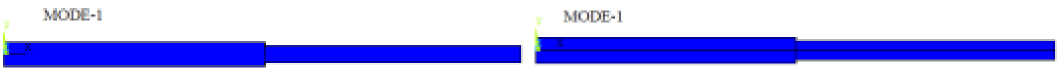


Fig :21 Mode-1 (Link 180 at left & Beam 188 at right)

ISSN PRINT 2319 1775 Online 2320 7876

Research paper

 UGC CARE Listed (Group -I) Journal Volume 8, Issue 1, 2019

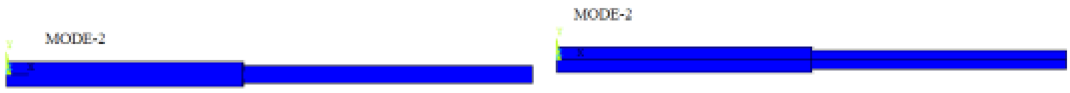


Fig :22 Mode-2 (Link 180 at left & Beam 188 at right)

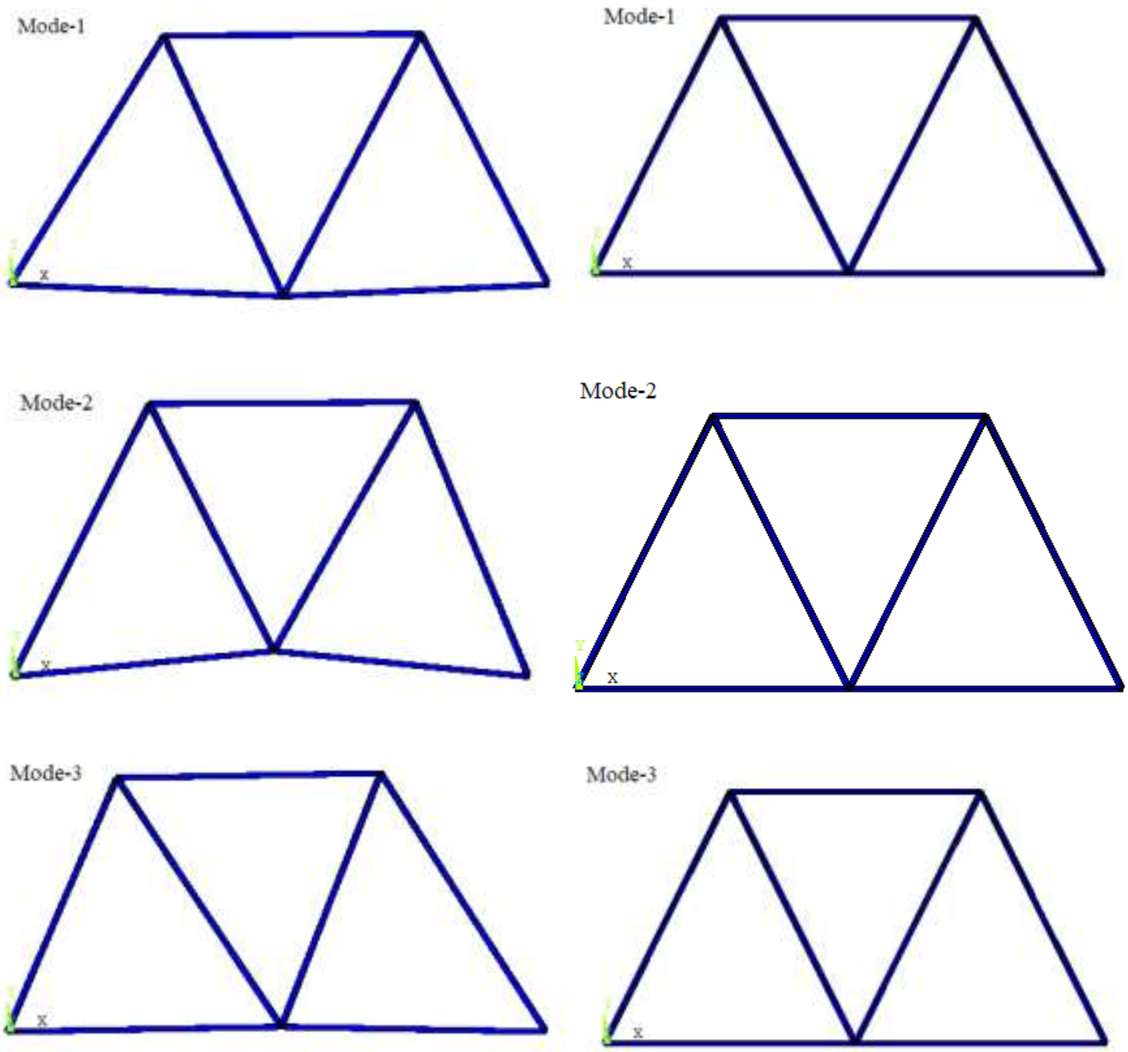


Fig :23 First Three Mode shapes of the truss structure with link 180 (left) and beam 188 (right) elements

ISSN PRINT 2319 1775 Online 2320 7876

 UGC CARE Listed (Group -I) Journal Volume 8, Issue 1, 2019

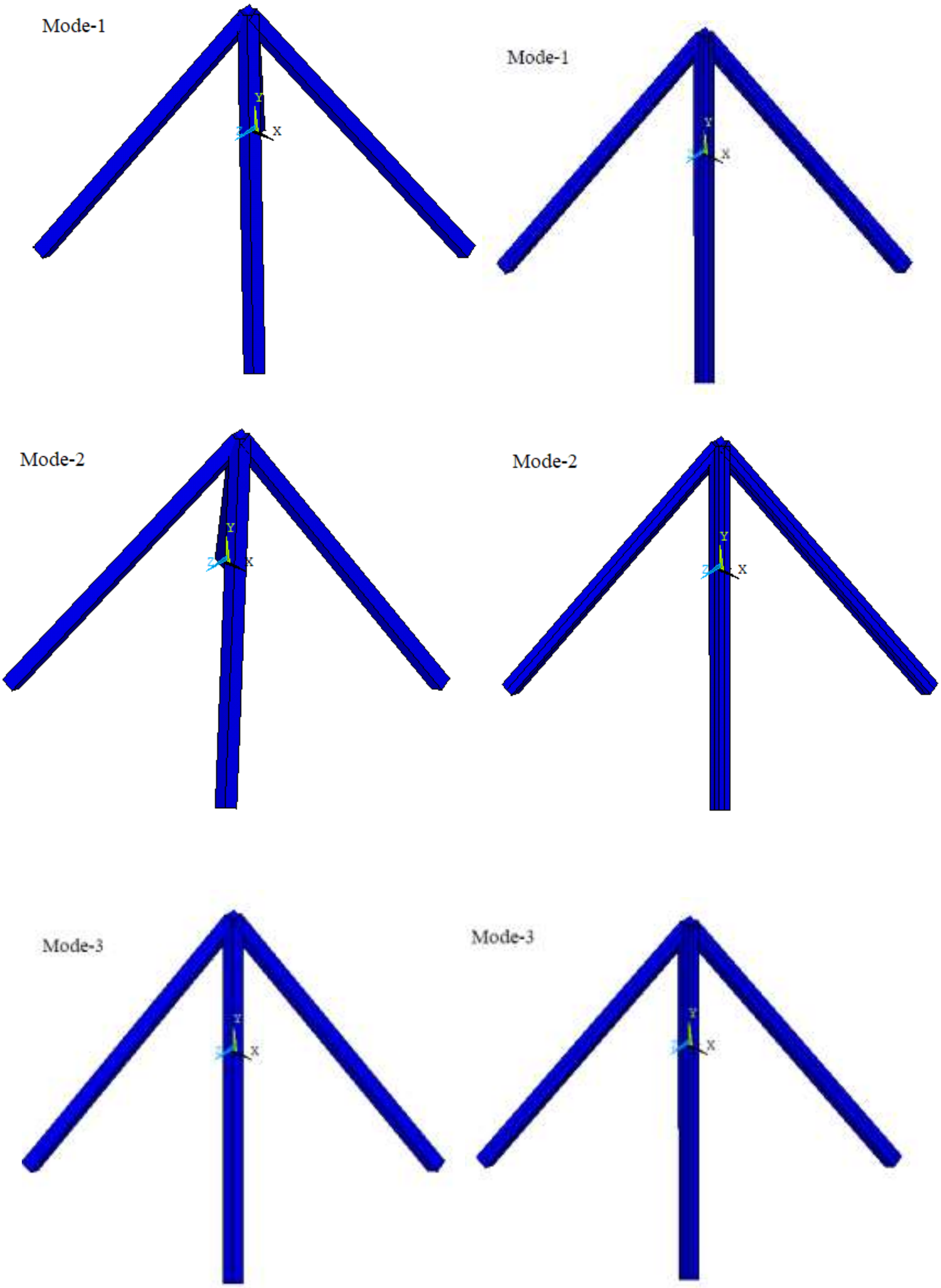


Fig :24 First Three Mode shapes of the space truss structure with link 180 (left) and beam 188 (right) elements

ISSN PRINT 2319 1775 Online 2320 7876

Research paper

 UGC CARE Listed (Group -I) Journal Volume 8, Issue 1, 2019

Table:4 Comparision of natural frequencies of Stepped bar

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

Table:5 Comparision 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

Table:6 Comparision 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.

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

ISSN PRINT 2319 1775 Online 2320 7876

 UGC CARE Listed (Group -I) Journal Volume 8, Issue 1, 2019

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(2012) The finite element analysis of truss structure based on ANSYS. Advanced Materials Research 594-597:2939-2944
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.(2014) Modern numerical simulation methods and its practical applications in engineering. Gongcheng Lixue/Engineering Mechanics, 31(4): 11-18.
9. Eswara Kumar A. et al.(2017) Dynamic analysis of flex seal of solid rocket motor nozzle”, Materials Today: Proceedings. 4(2):1590-1597.
10. ANSYS Element Reference Library 2017.
    Available at http://www.ansys.stuba.sk/html/elem_55/EBooktoc.htm.