IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
e-ISSN: 2278-1684, p-ISSN: 2320-334X
PP 13-18
www.iosrjournals.org

On Finite Element Analysis of Steel and RC Beams : Performance of Different Elements

Biswanath Charan[1], Pijush Topdar[2]
[1]*(M-Tech student, Dept. of Civil Engg NIT Durgapu,India)*
[2]*(Assistant Professor, Dept. of Civil Engg. NIT Durgapur, India)*

ABSTRACT :*Modelling a beam of either an isotrpic material like steel or a composite material like reinforced concrete (RC) having surface bonded fibre reinforced polymer (FRP) laminates for realistic prediction of structural behaviour is a challenging task. Most of the dedicated softwares for analysis and design of civil engineering structures lack the facility of modelling beams with additional layers of different material like FRP sheets. Therefore for modeling and analysis of such beams with surface- bonded FRP layers, it is necessary to choose an appropriate Finite Element (FE) software which is capable of modelling beams bonded with additional FRP layers. However, the first step towards such an effort is to properly model the virgin beam without FRP layers. In this context, choosing the appropriate element(s) is also an important task. Therefore In this paper, an effort is made to assess the performance of the relevant elements of a commercially available FE software for the above purpose. Numerical examples are solved for beams under various loading and boundary conditions; results are compared with available results, wherever possible, to assess the suitability of such elements and their range of applicability.*

Keywords- *Appropriate modelling, FE software (Abaqus), RCC beam, use of best suited finite elements. different boundary and loading conditions.*

1. Introduction:

Beams like any other components of a structural system undergoes deterioration over time. The reasons may vary from structural overloading to environmental effects. Very often it is impracticable to replace such components due to functional difficulties and economic reasons. Instead, retrofitting/strengthening of such deteriorated components is a viable alternative. There are many methods for strengthening. In this context, use of surface-bonded FRP layers for strengthening beams is one of the popular methods and it represents an interesting area of research. This technique has been widely investigated, andexamples of existing structures retrofitted using epoxy-bonded composite materials can be found in the literature [1–3].

While FE models for analysis of beams with surface-bonded FRP plates are not rare in the literature, use of commercially available FE software for such purpose is rare. On the other hand, the available commercial softwares for analysis and design of conventional civil engineering structures lack the facility of incorporating additional layers of different materials integrated to the virgin beam. However, for industrial purpose it is desirable that suitable software are made available to the analysts and designers along with proper guidelines to use them for appropriate modelling and analysis of beams with surface-bonded FRP plates.

To fulfill this requirement, a commercially available software ABAQUS-6.11 is chosen. However, as the first step towards this purpose, it is necessary to assess the suitabilty of the software for the simpler case i.e. modelling and analysis of the virgin beam without any additional layer.

Therefore, an effort is made in the present study to use the above software for modelling and analysis of beams made of steel and reinforced concrete (RC). In this context, choosing the appropriate element(s) out of the element library is extremely important for predicting the structural behaviour of the beam realistically with reasonable accuracy.

Consequently, various two and three dimensional elements are used for analysis of simply supported beams under different loading conditions for their suitability and range of applicability. Effect of varying the width of boundary strip and effect of different numerical integration scheme on the solution accuracy are also studied.

Results from the present analysis are compared with available results wherever possible. Some new results are also presented. The present study is limited to the linear analysis and only static cases are considered.

2. Elements Used for Analysis:

- 2 node linear beam element in plane–(designation B21 in Abaqus): It is an element use for plane stress analysis it has 3 degree of freedom at each node .It have two nodes. It is the simplest element in Abaqus and it is used for simple stress analysis [4]. Refer to Fig 1.
- 3 node quadratic beam in space-(designation B32): It is a quadratic element in space which have 6 degree of freedom at each node and it is used for stress concentration and for analysis of beam in space or frame in space [4]. See Fig 2.
- 8 node linear brick element reduced integration-(designation C3D8R): It is a8 node 3D brick element and it used for higher beam analysis it gives more accurate result. Displacements, rotations, temperatures, and the other degrees of freedom mentioned in the section are calculated only at the nodes of the element. At any other point in the element, the displacements are obtained by interpolating from the nodal displacements. Usually the interpolation order is determined by the number of nodes used in the element.Elements that have nodes only at their corners, such as the 8-node brick shown in Figure 18, use linear interpolation in each direction and are often called linear elements or first-order elements [4]. Refer to Fig3.
- 20 node quadratic brick element, reduced integration-(designation C3D20R): It is required for threedimentional (3D) section analysis and for design. Displacements, rotations, temperatures, and the other degrees of freedom mentioned in the previous section are calculated only at the nodes of the element. At any other point in the element, the displacements are obtained by interpolating from the nodal displacements [4] .See Fig 4.
- 3 node quadratic 3-D truss element-(designation T3D3): Here in the modeling of reinforced cement concrete beam the main reinforcement (tension reinforcement), compression reinforcement and stirrups are modeled by T3D3 elements [4].

3. Results and Discussion:

Various types of numerical examples of steel, concrete, RCC beams and RCC beam with FRPs lamination are presented. All examples carry out convergence studies for both analytical results and results getting from Abaqus software. Various elements, mesh sizes and plane conditions are applied for modelling. Different boundary strips with and loading conditions are applied in modelling.

3.1. Example 1:

This example is based on the convergence study of simple supported one end hinged and other end roller supported steel beam by different finite elements and mesh sizes in 2D plane. Comparing the result of simple supported centrally loaded isotropic plain beam in 2D plane i.e. 2-node linear beam element in plane (B21), 2-node linear beam in space and simply supported centrally loaded isotropic plain beam with 3-node quadratic beam in space (B32). Shown in Fig.5.

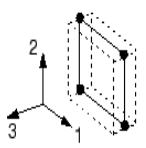
Material used steel grade Fe415, young modulus=200000 Mpa, Poisson's ratio= 0.28, density= 7.85e-5 N/mm^3 Dimension of beam: depth= 300 mm, width= 150 mm, effective span= 1500 mm. concentrated loading applied at middle of the span. The result shown in Table 1. Here we see that 3-node quadratic beam element gives better result for less number of meshes. And the result of deflection at mid node for 2-node linear beam element in plane at 400 number of elements converges with the magnitude of deflection at mid node for 3-node quadratic beam in space at 50 number of element at same loading.

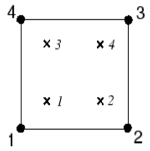
3.2. Example 2:

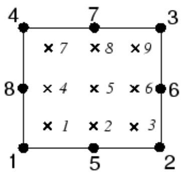
In this example the study is based on the 3D simply supported steel beam with linear load acting at center of the beam. The analysis is done both analytically and by software. In this discussions different strip width are used as boundary condition. Loading as linear with 1mm strip width with 20 node quadratic brick element reduced integration (C3D20R) and 8 node linear brick element reduced integration (C3D8R) in individual cases (3D beam element) with varying mesh sizes. Material used: steel grade Fe415, young modulus=200000 Mpa, Poisson's ratio= 0.30, density= 7.85e-5 N/mm^3. The modelling shown in Fig 6. The results are shown in Table 2. Here we see C3D20R gives much better result compare to C3D8R element with suitable mesh size. The results of maximum deflection at the middle node of middle element of lower surface given by Abaqus with analytical results differ with variation of elements and mesh sizes. From result we see that C3D20R element at 30 mm mesh size (loading applied at middle of the span as linear 1mm strip) gives less error compare to others, the percentage of error 7.11 and C3D20R at 20 mm mesh size (loading applied at middle of the span as linear 10 mm strip) gives same result as simple beam in 2D plane (B21, B32) example1. That why these analysis are done to see which element at which mesh size results are converged.

3.3. Example 3:
Earlier problems deal with one isotropic material but in this example reinforced concrete beam is modelled and compare the result of deflection and misses stresses. RCC beam, dimension 100×150×1200 mm. Grade of concrete M20, with tension reinforcement - 2 no@10mm tor bar and compression reinforcement – 2 no.@8mm tor bar Fe 415 grade used .For stirrup 6mm tor bar used @ 100mm c/c. Refer to Fig.7. **Concrete** :(a) Material : M20 concrete, young modulus(E)=22360.679 Mpa, Poisson's ration ($\boldsymbol{\upsilon}$)=0.17,elastic material taken, density=2.5*10^{-5}N/mm^3 (b) Element: 20 node quadratic brick element, reduced integration – C3D20R **steel**:(a) Material : Fe 415 grade steel , young modulus(E)=200000Mpa, Poisson's ration ($\boldsymbol{\upsilon}$)=0.30,elastic material taken. Density =7.85*10^{-5}N/mm^3(b) Element: 3 node quadratic 3-D truss element – T3D3.**Boundary condition:** Simply supported beam one end hinge supported and other end roller supported. Supports are 100 mm apart from beam ends. Strip of 15 mm is applied to setup support. Loading**:** Two strip loads here assume 5mm strip as loading is acted from equal distance from the supports .Two equal linear loads each acted 325 mm from the support. Results are given in Table 3 and Table 4. The load vs deflection graph of mid span middle element for RCC beam is shown in Fig. 8.

4. Figures and Tables:







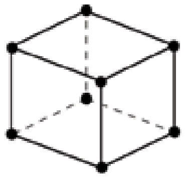


Fig.1: 2node linear beam, B21

Fig.2: 3node quadratic beam, B32

Fig.3: 3node linear, C3D8R

Fig.4: 20node, C3D20R

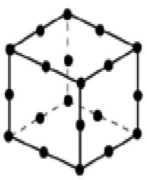
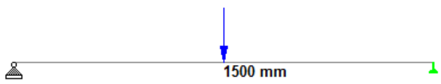


Fig.5: simple supported beam



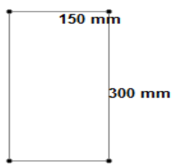
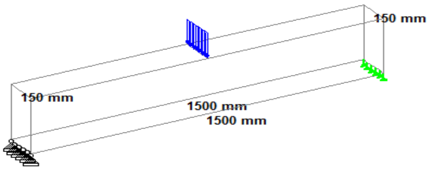


Fig.6: 3D Beam

Fig.7: RCC beam



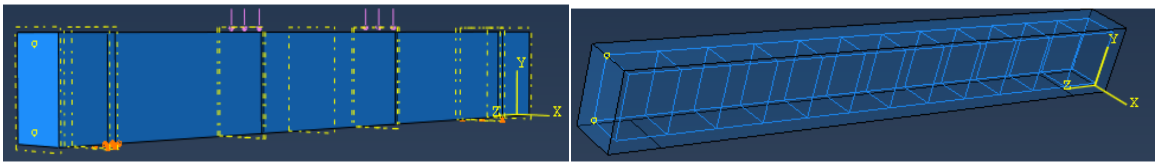


Fig.8: load vs deflection graph of RCC beam

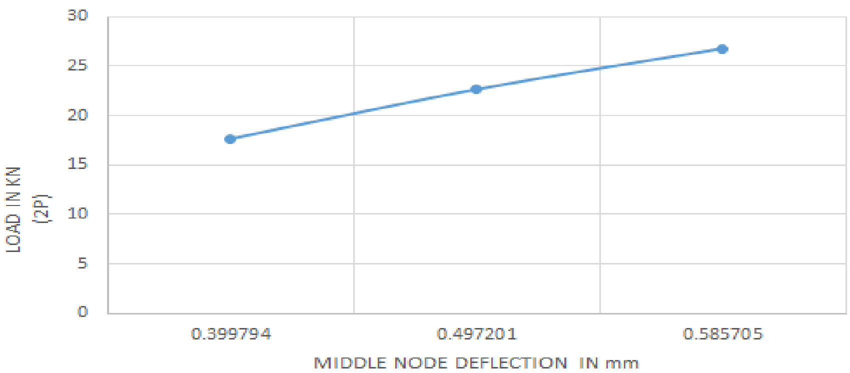


TABLE 1: Study on Results of Simply Supported Beam in 2D Plane (2node Linear Beam Element in Plane & 3-Node Quadratic Beam in Space)

ELEMENT TYPE	LOAD AT CENTRE (KN)	MESH SIZE(number of element)	DEFLECTION AT CENTRE POINT BY ABAQUS(MM)	ANALYTICAL DEFLECTION AT CENTRE(MM)	% ERROR/ VERIFICATION
A-2 NODE LINEAR BEAM ELEMENT IN PLANE	10	400	0.0116912	0.0104166	12.23
	100	400	0.116711	0.104166	12.23
A-3 NODE QUADRATIC BEAM IN SPACE	10	50	0.0116912	0.0104166	12.23
	100	50	0.116711	0.104166	12.23

TABLE 2: Study on 3D Simply Supported Steel Beam (3D Brick Element)

BOUNDARY CONDITION (as simply supported)	LOADING (line loading /uniform pressure) 100 KN	MESH SIZE	ELEMENT TYPE	CENTRAL MAXIMUM DEFLECTION BY ABAQUS	ANALYTICAL MAXIMUM DEFLECTION AT CENTRE	% ERROR (+VE)
10 mm STRIP	LINEAR 1MM STRIP	25 mm	A-20 NODE QUADRATIC BRICK ELEMENT REDUCED INTEGRATION	0.114919	0.104166	10.32
15 mm STRIP	LINEAR 1MM STRIP	30 mm		0.111576		7.1136
10 mm STRIP	LINEAR 1 mm STRIP	50 mm		0.114117		9.55
10 mm STRIP	LINEAR 5 mm STRIP	15 mm	A-20 NODE QUADRATIC BRICK ELEMENT REDUCED INTEGRATION	0.117848		13.12
15 mm STRIP	LINEAR 10 mm STRIP	20 mm		0.116690		12
10 mm STRIP	LINEAR 5 mm STRIP	15 mm	AN-8 NODE LINEAR BRICK ELEMENT REDUCED INTEGRRATION	0.122744		17.83
15 mm STRIP	LINEAR 10 mm STRIP	20 mm		0.119268		14.49
20 mm STRIP	LINEAR 10 mm STRIP	25mm		0.117766		13.05

TABLE 3: Load Deflection Results by Abaqus Software of RCC Beam

BEAM DESIGNATIN	LOADING(2P)IN KN	LOWER SUFFACE MIDDLE NODE MAX. DEFLECTION OF BEAM (mm)	EXPERIMENTAL MAX. DEFLECTION OF BEAM (mm)
RCC BEAM	17.68	0.399794	0.48
	22.68	0.497201	0.57
	26.8	0.585705	0.71

TABLE 4: Misses and Flexural Stresses Results by Abaqus Software of RCC Beam

BEAM DESIGNATIN	LOADING(2P)IN KN	LOWER SUFFACE MIDDLE ELEMENT STRESSES OF BEAM WITHOUT FRP LAMINATE(N/mm^2)		
		S_{11}	S_{22}	S_{33}
RCC BEAM	17.68	5.15639	4.50234	5.15626
	22.68	6.62464	5.78563	6.61448
	26.8	7.82625	6.84481	7.82605

5. Conclusion:

The behaviour Of RCC beam, pure steel beam and other structural beams are studied by using Finite Element Software ABAQUS6.11. Consequently model elements-B21, B32, C3D8R, C3D20R, T3D3 are used for analysis purpose. Numerical example are carried out to study the performance and the range of applicability of these elements. For these purpose, RCC beam with various elements, various boundary support (strip width) conditions, various mesh size are considered under different loading. The result are compared with available solutions and theoretical results, wherever possible. For each case near about 8-12% error occurred for case of B21,B22,B32,B31 element and 6-8 % error occurred for elements C3D20R,T3D3,M3D8R.Different element gives different approximation. Varying mesh gives different approximation. Full integration gives more accurate result than reduced integration. Reduced integration gives over estimation. Here we see that ABAQUS software gives approximation near to the analytical as well as experimental results in static linear state. In modelling of RC beam there, within the range of elastic zone, the misses stresses are very small compare to yield stress. And load vs deflection graph approximately linear.
Form the various modelling here, the results tailed with analytical and experimental results. The software gives good results. So we can proceed to the next step that is modelling and analysis of strengthening and retrofitting of RCC beams by FRP layer by this software.

References:

[1] [1-3]G.M. Chen a, J.F. Chen b, J.G. Teng,*On the finite element modelling of RC beams shear-strengthened with FRP*, Construction and Building Materials 32 (2012) 13–26.
[2] Francois Buyle-Bodin , Emmanuelle David , Eric Ragneau,*Finite element modelling of flexural behaviour of externally bonded CFRP reinforced concrete structures,*Engineering Structures 24 (2002) 1423–1429.
[3] Ehab Ellobody, *Performance of composite girders strengthened using carbon fibre reinforced polymer laminates*, Thin-Walled Structures 49 (2011) 1429–1441 (Contents lists available at ScienceDirect).
[4] Z.J. Yanga,, J.F. Chenb, D. Proverbsc, *Finite element modelling of concrete cover separation failure in FRP plated RC beams,* Construction and Building Materials 17 (2003) 3–13.
[5] ABAQUS 6.11 documentation & ABAQUS element library

Books:

[1] “Strength of Materials” by R.S. Khurmi
a. “Theory of Structures” by S. Ramamrutham and R. Narayan.