ISSN: 2348 - 2117

International Journal of Engineering Technology and Computer Research (IJETCR)
Available Online at www.ijetcr.org
Volume 5; Issue 3; May-June: 2017; Page No. 22-40
Journal Approved by UGC

Non-Linear Finite Element Analysis of Steel Plate Retrofitted RC Beam Using ANSYS

Nandhu Prasad[*1], Ramveer Tyagi[2], John Hansdak[3]

[*1]*Integrated M.Tech Student*, Department Of Civil Engineering, Dr.K.N.Modi University, Newai, Rajasthan, India.*

[2]*Associate Professor and Head, Department of Civil Engineering, Dr.K.N.Modi University, Newai, Rajasthan, India.*

[3]*Associate Professor, Department Of Civil Engineering, Dr.K.N.Modi University, Newai, Rajasthan, India.*

Abstract

The objective of this paper is to study the behaviour of retrofitted and non-retrofitted beams and compare the results. The retrofitting material used here is the steel plate, which is attached to the soffit of the beam. A 3-dimensional nonlinear finite element analysis of the beam subjected to single point loading has conducted. Different concrete failure parameters such as shear transfer coefficients, uniaxial tensile and compressive strengths considered. Full-scale models of control beams, as well as retrofitted beams, are shaped and results compared. The first crack load, failure load, crack pattern and load deflection characteristics of the beams modelled and analysed using finite element analysis package compared with the experimental results. The finite element analysis technique in this present study is carried out using ANSYS 15 Package. The various elements employed in ANSYS to model the reinforced concrete beam, as well as the different inputs required, along with the steps to be followed to achieve the results is detailed in this paper. Conclusions have been drawn out by comparing the results of the Control and Strengthened beam. The steel plate retrofitted beam was seen to have enhanced strength and also compared well with the experimental data.

Keywords: Non linear analysis, Finite Element, Steel plate, Retrofitting, Structure, Strengthened beam, Full Beam, Control beam

I. Introduction

In the profession of structural engineering, new contemporary researches were carried out using advanced materials to improve structures considering strength aspect. Due to innovations the plain cement concrete was prefaced with steel members and it gives quite satisfying results, but the problem is that the destructive steel member added in the plain cement concrete may get corroded if it is affected by moisture content. It can be overcome by new ideas that emerged, and one such kind is retrofitting. Retrofitting can apply on old structures, and structures in the seismic zone to resist their structural collapse. Retrofitting means the changes that will happen on anything after it was manufactured. Using composite materials, we can make Retrofitting. We can enhance the strength of existing structures against seismic activity by doing retrofitting process.

The constituents of the composite materials have different physical and chemical properties. However, when they fuse together, the last material has distinctively different characteristics. Even after the combination, the constituents remain separated in the material. The technologically most important composite are those in which the dispersed phase is in the form of fibre. The fibres are either long or short. Long and continuous strands are easy to orient and process, whereas short fibres cannot be controlled adequately for proper orientation. The principal in commercial use is various types of glass, carbon, graphite and Kevlar. All these fibres incorporated in matrix form either in the continuous length or discontinuous length. Polymers like polyester, vinyl ester or nylon are used even though it is usually epoxy. The method of preparation, the relative proportion of fibres to the polymer and the layout determines the properties of FRP.

Concrete elements are traditionally performed by externally bonding steel plates to concrete as retrofitting of flexures. This process is very efficient in increasing the strength and stiffness of reinforced concrete elements, and it has the limitations of being susceptible to corrosion and is difficult to install. Latest development in the field of composite materials, together with their inherent properties, which include high specific tensile strength real fatigue and corrosion resistance and ease of use, steel plates make them an attractive alternative in the field of repair and strengthening of concrete elements

II. ABOUT ANSYS

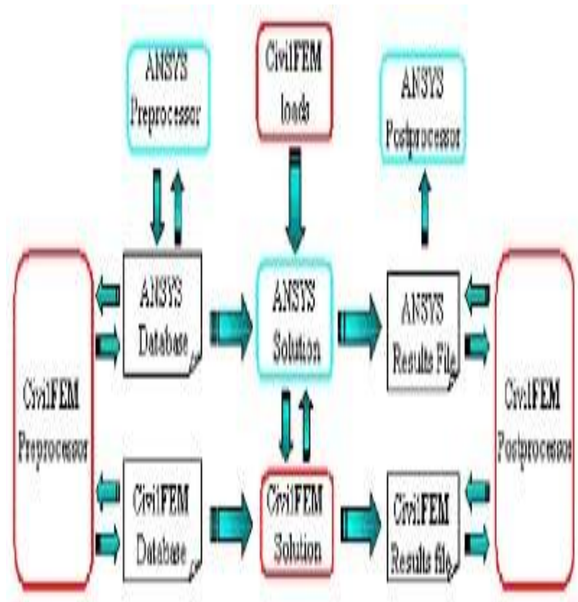
ANSYS is designed simulation software (computer-aided engineering, or CAE). In 1996, the NASDAQ stock exchange listed ANSYS. Investor's Business Daily ranked ANSYS in late 2011 as one of the only six technology firms universal to receive the highest possible score on its Intelligent Select Composite Ratings.

Finite element analysis as a tool is mainly used to verify the sections tested because, these sections being thin-walled and having perforations throughout the length, their behaviour is quite complicated when subjected to axial loads. Shell elements available in ANSYS [20] software gives a real means to verify the experimental results.

ANSYS have been recognised as a high performer by many other sources as well. The organisation reinvests 15 percent of its revenues each year into research to continually refine the software. ANSYS offers a broad range of engineering simulation solution which will be providing access to almost any area of engineering simulation that a design process requires. Companies in the diversity of enterprises practice ANSYS. The tools put a useful product through an inflexible trial method (such as crashing a vehicle into a masonry wall, or running for several years on a tarmac road) before it becomes a physical object.

The skeletal integrity and stability of any building are only as good as its each part. The way those parts fit mutually, along with the choice of materials and the construction site, all give to how the building will do under normal — or extreme — conditions. Civil engineers integrate this knowledge into their building designs and comply with increasingly demanding safety and government regulations.

Civil Engineers use ANSYS software for designing projects which are diverse as high-rise buildings, bridges, dams, stadiums, etc. by testing with innovative design in a virtual environment, dams, and stadiums. Engineers and designers can analyse safety, strength, comfort and environmental consideration.

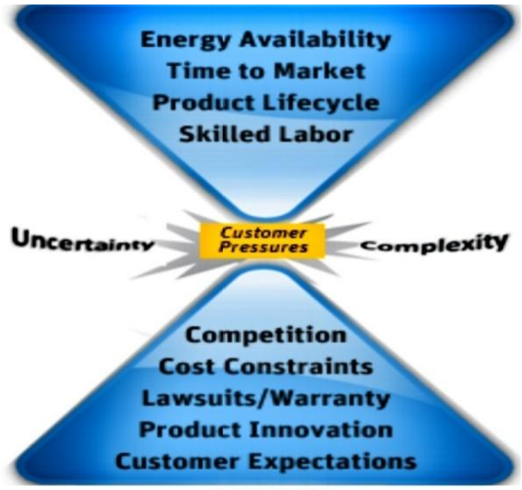


PROCESSING OF ANSYS CIVIL

2.1 ANSYS Products

Computer Modelling Technology: Structural Mechanics, Multi-physics, Fluid Dynamics, Explicit Dynamics, Electromagnetism. Workflow Technology: ANSYS Workbench Platform, High-Performance Computing, Geometry Interfaces, Simulation Process & Data Management. Practically every industry now recognises that a key strategy for success is to merge computer-based design simulation early in the evolution process, allowing designers to refine and validate designs at a stage where the cost of making changes is least. At ANSYS, we bring clarity and insight to customer's most complex design challenges through fast, right and reliable simulation. Our technology enables organisations to predict with confidence that their products will thrive in the real world. They trust our software to make sure the product integrity plus drive marketing achievement through discovery. Every product is a promise to live up to and surpass expectations. By simulating early and often with ANSYS software, our customers become faster, more cost-effective and more creative, realising their product bonded layers of orthotropic materials.

In the nonlinear analysis, due to bending, different sandwich layers will be in various situations of strain. So, a layered access can be adopted assuming the sandwich layer to have constant anxiety across its thickness even though the stress varies in the thickness direction of the whole lamina.



ANSYS

III. MATERIAL INVESTIGATION

Retrofitting of flexural concrete elements is traditionally fulfilled by externally bonding steel plates to concrete. Still, this method has proved to be powerful in increasing stiffness and strength of reinforced concrete components; it has the disadvantages of being susceptible to corrosion and difficult to install. Recent development in the field of composite materials, together with their inherent properties, which include high specific tensile strength real fatigue and corrosion resistance and easiness of use, make them an attractive option to steel plates in the field of repair and strengthening of concrete elements.

3.1 Steel plates

There are different techniques in retrofitting such as steel jacketing, providing additional reinforcement, increasing the section dimensions and strengthening using fibre reinforced polymers. Each of these methods has their importance and is opted based on the requirement and ease of application. The above techniques are expensive, require skilled labour and take more duration for use. Mild steel plates will be one of the alternatives for retrofitting the existing structure, which has high young's modulus and is ductile, malleable. They are reasonable in price and easy to install. They will be available in required thickness. They can be painted along with the specimen so that it will not appear as retrofitted specimen.

- Advantages of steel plates:
  - Causes minimal site disruption
  - Minimal change in section size
  - Quick

- Disadvantages of steel plates:
  - Difficulty in handling heavy plates
  - Corrosion

RCC Beam retrofitted with MS plates

3.2. Fibre-Reinforced Polymers (FRP)

Reinforcing Fibre is available in both two-dimensional and three-dimensional orientations

1. Two-dimensional Fibre-Reinforced Polymers characterised by a laminated structure in which they are aligned along the plane in x-direction and y-direction of the material. Nothing has aligned through thickness or the z-direction, this lack of arrangement along the thickness can create a lack of payment plus processing. Cost and labour increase because traditional processing methods used to make composites, such as wet hand lay-up, autoclave and resin transfer frame, require a high amount of skilled labour to cut, stack and consolidate into a preformed component.

2. Three-dimensional Fibre-Reinforced Polymer composites are materials with three-dimensional fibre structures that incorporate in the x, y and Z-directions. The development of 3-dimensional orientations arose from industry's need to reduce fabrication expenses, to increase through-thickness

mechanical characteristics, and to improve impact damage tolerance; all were problems associated with two-dimensional fibre-reinforced polymers.

FRP allows the alignment of the glass of thermoplastics to suit specific design programs. Specifying the orientation of reinforcing can increase the toughness and invulnerability to deformation of the polymer. Glass reinforced polymers are compact and most resistive to deforming energies when the polymers are parallel to the force exerted and are weakest when they are perpendicular. Thus this ability is at once both an advantage and a limitation depending on the context of use. Weak spots of perpendicular can be utilised for natural hinges and joints, but can also lead to material breakdown when production processes fail to orient the parallel to conventional forces adequately. When forces are exerted perpendicular to the orientation of the strength, the elasticity of the polymer is less than the matrix alone.

Structural collapse can occur in FRP materials when:

- Tensile forces stretch the matrix more than the limit, causing the material to shear at the interface.
- Tensile forces near the end of the fibres exceed the tolerances of the matrix, separating the fibres from the matrix.
- Tensile forces can also exceed the tolerances of the fibres causing them to fracture leading to material failure.

FRP retrofitted beam

➢ Advantages of FRP
- Light weight, durable, economical.
- Thin laminates.
- Chemically inert.
- High strength to weight ratio.

➢ Disadvantages of FRP
- Mismatch in tensile strength with that concrete.

Since its tensile strength reduces with rising modulus, its strain at fracture will also be much lower. Because of the element brittleness at higher modulus, it becomes critical in joint and connection details, which can have high-stress concentrations. As a result of this phenom, carbon composite laminates are more efficient with the adhesive bonding that eliminates mechanical fasteners.

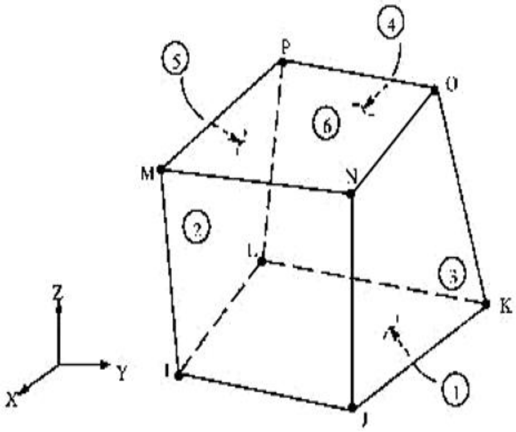
IV.FINITE ELEMENT ANALYSIS

Finite Element Analysis (FEA) is a kind of computer application that uses the finite element method to analyse a material or object and find how applied stresses will affect the materials or plan. FEA can help resolve any points of weakness in a design before its manufacture. FEA programs are more broadly available with the scope of more powerful computers but are still wildly used in aerospace and other high-stress applications. The analysis should have to do by generating a mesh of points in the mould of the object that contains knowledge about the element and the object at each point for review. In addition to learning the reaction to stress upon an object, Finite Element Analysis can also analyse the effect of vibrations, fatigue, and heat transfer.
The beams that selected for modelling simply supported and loaded with centrally placed concentrated transverse loads. The analysis for the reinforced concrete beam included: non-linear real properties, a linear bond-slip relation, bilinear steel properties, and the influence of progressive cracking of the concrete. The transverse loading was incrementally applied and ranged in magnitude from zero to a load well above that which initiated cracking.

4.1 ELEMENTS USED FOR DISCRETISATION

4.1.1 Concrete

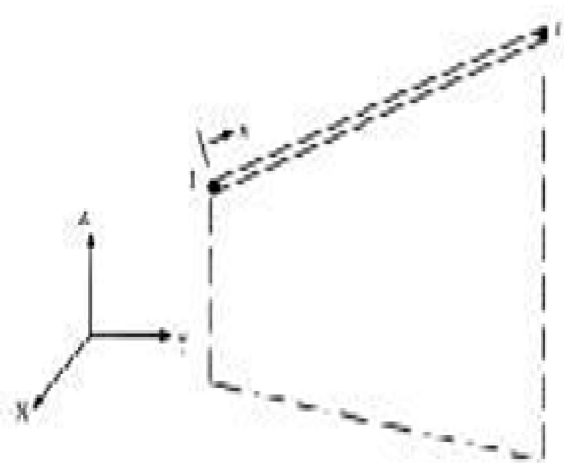
An eight-node solid element, Solid65, was used to model the concrete. The solid element has eight nodes with three degrees of freedom at each node – translations in the nodal x, y, and z directions. The element is capable of plastic deformation, cracking in three orthogonal directions, and crushing. The geometry and node locations for this element type shown in Figure



SOLID 65

4.1.2 Steel Reinforcement

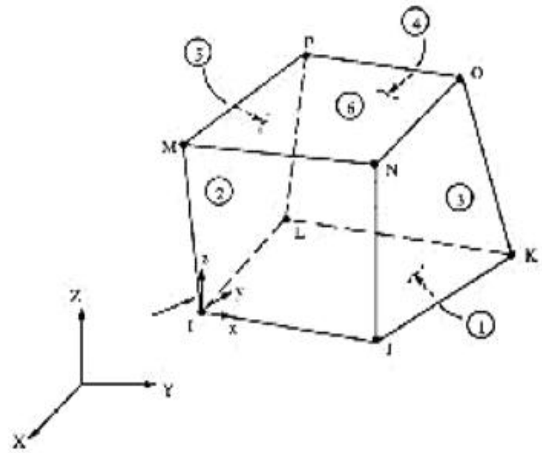
A Link180 element was used to model the steel reinforcement. Two nodes are required for this item. Each node has three degrees of freedom, – translations in the nodal x, y, and z directions. The element is also capable of plastic deformation. The geometry and node locations for this element type are shown in Figure.



LINK 180

4.1.3 Steel Plates

An eight-node solid element, Solid185, was used for the steel plates at the supports in the beam models. The element defined with eight nodes having 3-degrees of freedom at each node – translations in the nodal x, y, and z directions. The geometry and joint locations for this element type shown in Figure



Solid185 – 3-D solid(ANSYS 2015)

4.2 CONSTITUTIVE MODELLING OF CONCRETE

Cement is a brittle material and has different behaviour in compression and tension. In compression, the stress – strain curve relationship for concrete is described by multi- linear isotropic curve, linear elastic up to about 40 percent of the maximum compressive strength (FC'). After it reaches the maximum compressive strength, the curve descends into softening region, and eventually, crushing failure occurs at an ultimate strain. In tensioning upon concrete, the stress-strain curve is relatively linear elastic up to the maximum tensile strength, after which concrete cracks and strength decrease to zero.

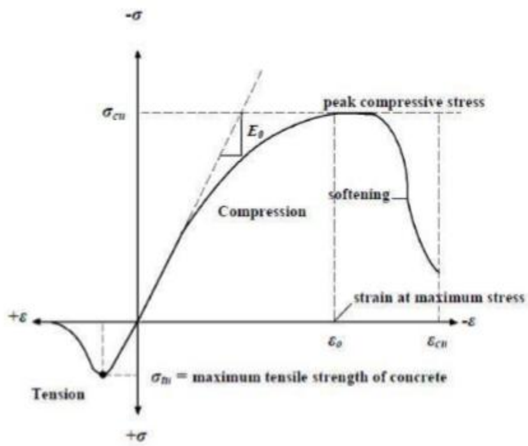
4.3 Stress-Strain Relation for Concrete

$$f = \frac{E_c \varepsilon}{1+(\frac{\varepsilon}{\varepsilon_c})^2}$$

$$\varepsilon = \frac{2f'_c}{E_c}$$

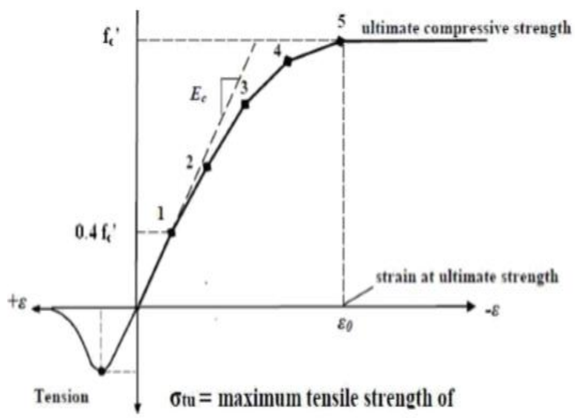
$$E_c = \frac{f}{\varepsilon}$$

Where f = stress at any strain ε, MPa
ε = strain at stress f
εc = strain at the ultimate compressive strength FC



Uniaxial Stress-Strain curve for concrete

The simplified stress-strain curve constructed from six points connected by straight lines. The strain at point 1 is 0.4 FC' and points 2,3,4,5 are obtained using the equations 1-3. The last point is made entirely plastic. Poisson's ratio of concrete is 0.2



Simplified compressive uni-axial stress strain curve for concrete

4.4 STRESS STARIN RELATIONSHIP OF STEEL

Steel reinforcement in the experimental beams constructed with typical Grade 60 steel reinforcing bars. Properties, i.e., elastic modulus and yield stress, for the steel reinforcement used in this FEM study follow the design material properties used for the experimental investigation (Kachlakev and McCurry 2000). The steel for the finite element models was assumed to be an elastic-perfectly plastic material and identical in tension and compression.

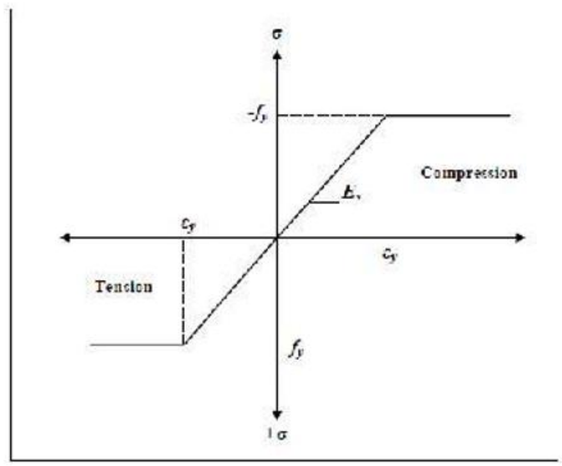
Poisson's ratio of 0.3 used for the steel reinforcement in this study (Gere andTimoshenko1997). The figure shows the stress-strain relationship utilised in this study. Material properties for the steel reinforcement for all four models are as follows:

Elastic modulus, Es = 200,000 MPa (29,000 psi)

Yield stress, fy = 413.4 MPa (60,000 psi)

Poisson's ratio, v= 0.3

Steel plates were added at support locations in the finite element models (as in the original beams) to provide a more even stress distribution over the support areas. An elastic modulus equal to 200,000 MPa (29,000 psi) and Poisson's ratio of 0.3 used for the plates. The steel plates were assumed to be linear elastic materials.



Stress-strain curve for steel reinforcement

4.5 Input Data

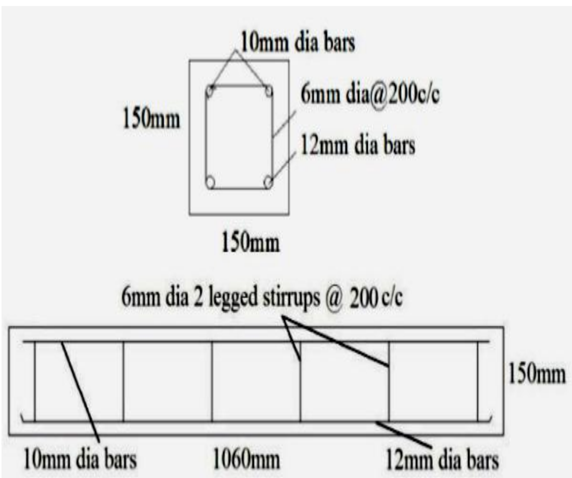
The Young's Modulus, Poisson's Ratio, Yield Stress, Density, etc. of the steel, concrete and the material used for retrofitting should give as the input data. Also, the particular specimen's (in one of the model) and the composite that is carbon fibre layered particular sample's (in the retrofitted and the rehabilitated models) face on which the uniformly distributed force is working will be given as an input. The boundary conditions such as fixity, to be introduced at the ends of the columns are also to give.

4.6 Output Information

The output involves strains, stresses, reaction forces, translations and moments. Segment forces, moments, and transverse shear forces are available for elements with displacement degrees of freedom. The nodal displacements, the support reactions and the average stresses σx, σy, and σz and the shear stress such as τxy, τyz and τzx as well as the common strains such as εx, εy, εz and the shear strains such as γxy, γyz and γzx are to obtain. The displacement in all the directions that is x,y and z directions are obtained. Moreover, lastly, a contour plot of all the output parameters mentioned above is obtained.

V. FINITE ELEMENT MODELLING OF RCC BEAM

Finite element modelling comprises of using an idealised element and meshing of items to replicate the RC beam. A perfect bond considered between the concrete and steel reinforcement. The RC beams utilised in the analysis were directly supported and strengthened with outer steel plates attached to their bottom.



Reinforcement detailing of the beam

5.1 General Details

28-day concrete strength =30MPa
Yield strength of reinforcing steel =415MPa
Grade of steel cover to reinforcement =415 MPa
(Flexural and stirrups) =25mm

5.2 Steps in ANSYS Modelling

First of all, the preference is set to structural. Analysis in ANSYS consists of 3-Steps

1. **Pre-processor**
- Element type can be beam/ Solid Model/ Heat Element/ Electromagnetic Element
- Real constant Here area, thickness are defined
- Material different properties elastic, inelastic etc are defined
- Section
- Modelling
- Meshing
- Loads and Boundary conditions

2. **Solution**
- Involves giving the analysis type, defining boundary conditions, assigning load, solving the problem

3. **General post processing**
- From where results are read, tables are listed, graphs are plotted

5.3 Element Types

Concrete	Solid 65
Rebar	Link 180
Steel Plate	Solid 185

Element Types

VI. ANALYTICAL RESULTS

The ANSYS application records a crack pattern at each applied load level. A cracking sign represented by a circle appears when a principal tensile stress exceeds the ultimate tensile strengths of concrete. The cracking sign appears vertically to the direction of the principal stress. ANSYS program displays circles at locations of cracking or crushing in concrete elements. Cracking shown with a circle outline in the plane of the crack, and crushing shown with an octahedron outline. The first, second and third cracks at an integration point are demonstrated by a red, green and blue circle outlines respectively. The ANSYS provides a Crack/Crushing plot option which shows the cracking pattern in the beam. To see the cracking in the model, turn on the Vector Mode plots. Flexural cracks form vertically up the beam. Compression failures will show as circles beneath the load. Diagonal tension cracks form diagonally up the beam towards the loading that is applied. In general, flexural cracks occur early at midspan. When applied loads increase, vertical flexural cracks spread horizontally from the midspan and reaches the support. At a higher applied load, diagonal tensile cracks appear. Increasing applied loads induces additional diagonal and flexural cracks. Finally, compressive cracks appear at nearly the last applied load steps. The crack patterns, deflections, at first crack, yielding point and Ultimate load shown in the following figures.

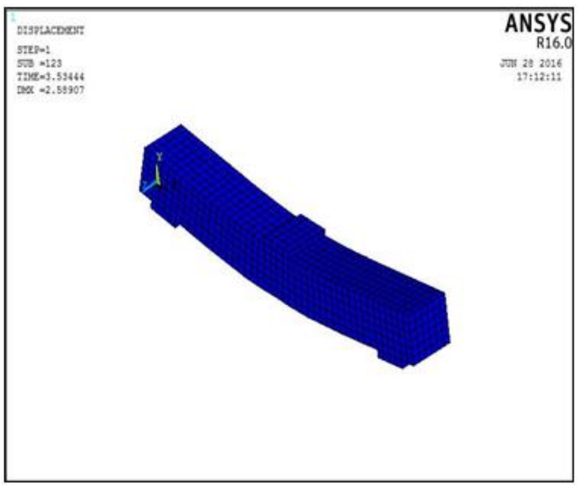
6.1 Control Beam And Retrofitted Beam

For the control beam, the first crack appears for the full beam. The cracks first occurred in the no shear span indicating a flexural crack. The cracks later extended upwards. With the increasing in load steps, the diagonal compression cracks also began to show the yielding of the steel.

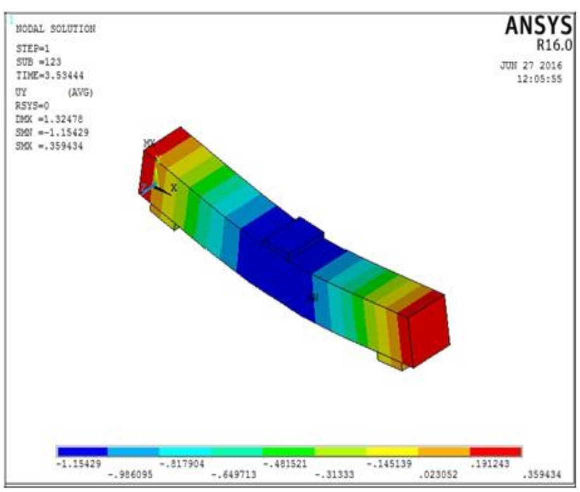
Failure of the beam was observed when the solution failed to converge for even more no. of substeps.ie when the load increment is minuscule.

FULL BEAM

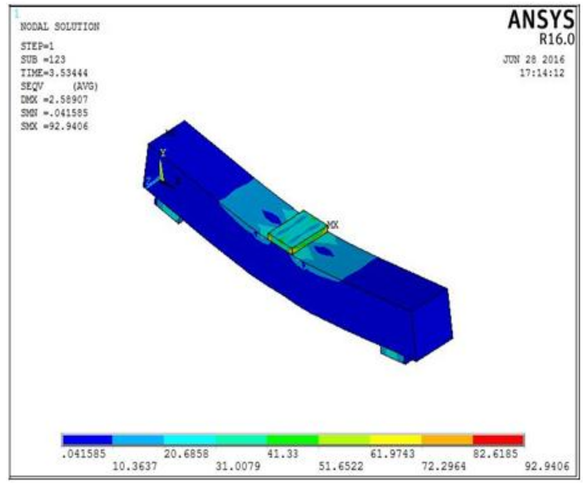
Control beam:



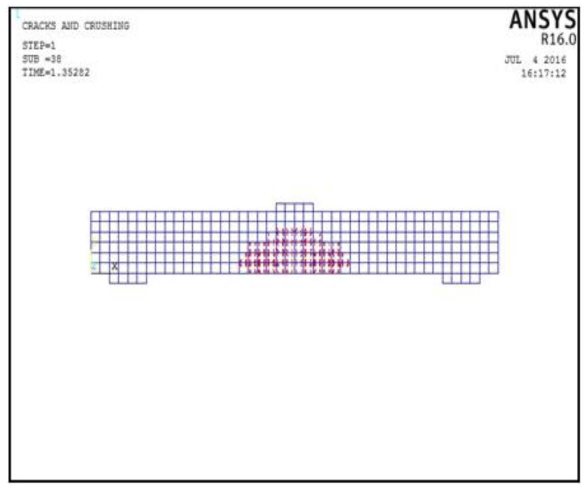
Deformation pattern at ultimate load



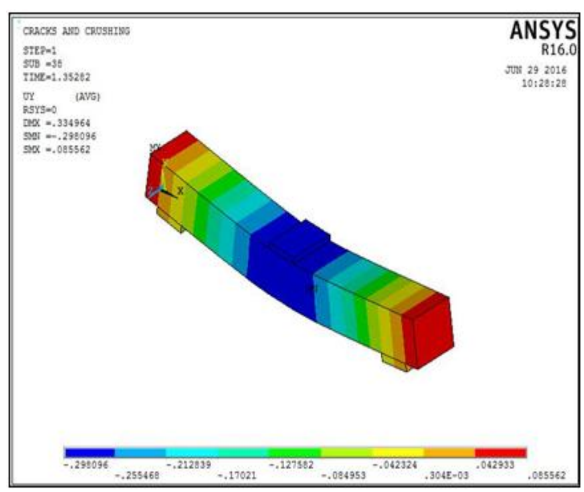
Deflection contour at ultimate load



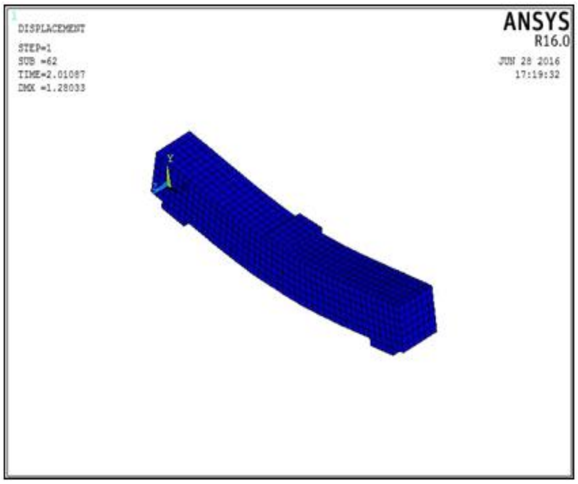
Stress contour at ultimate load



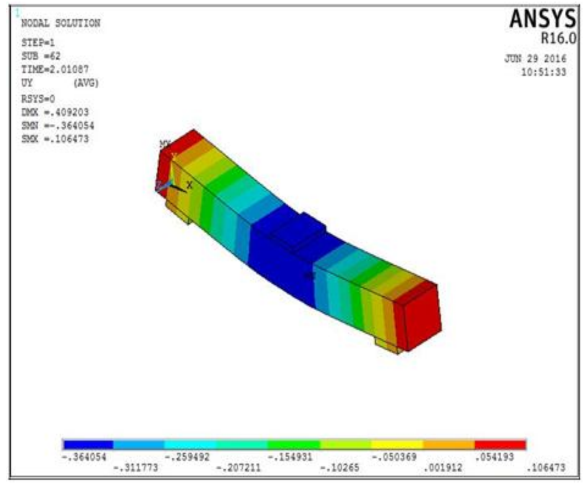
First crack



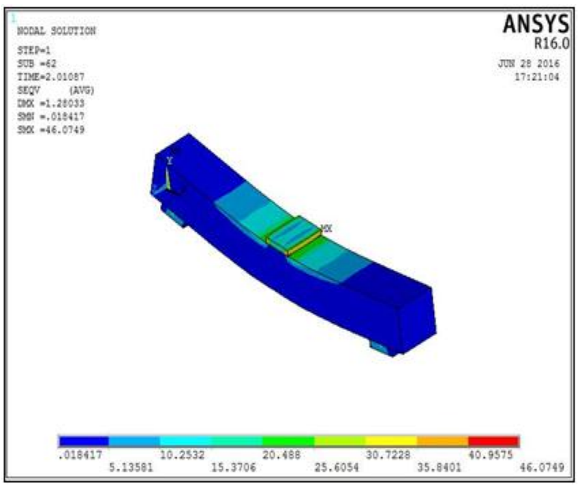
Deflection contour at first crack



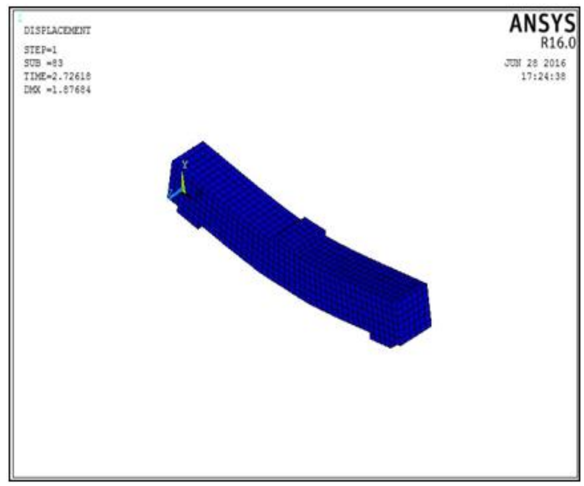
Deformation pattern at 30 KN



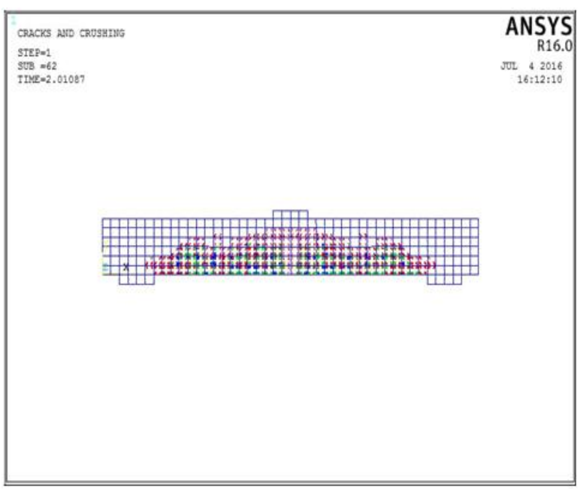
Deflection contour at 30 KN



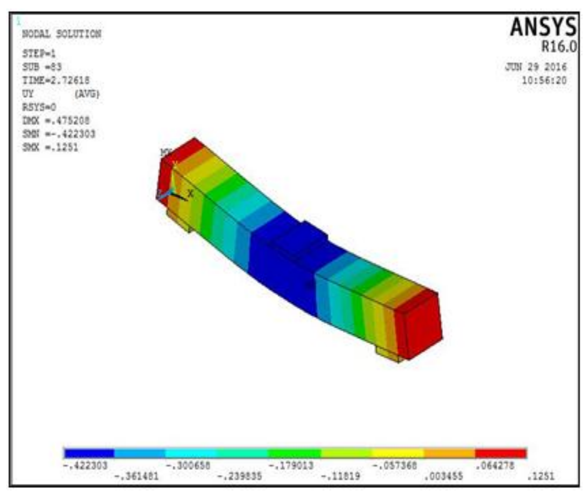
Stress contour at 30 KN



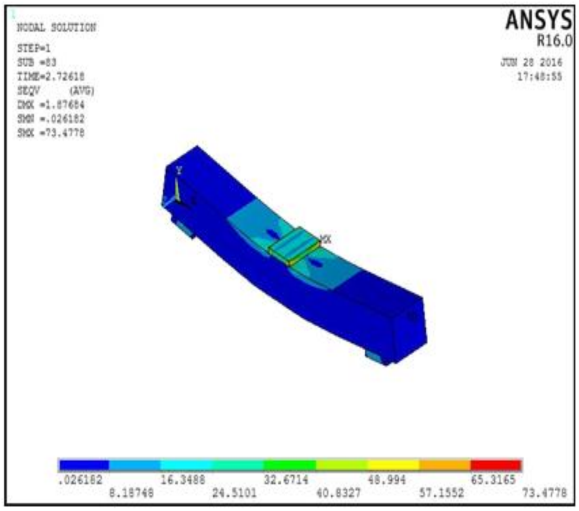
Deformation at 40 KN



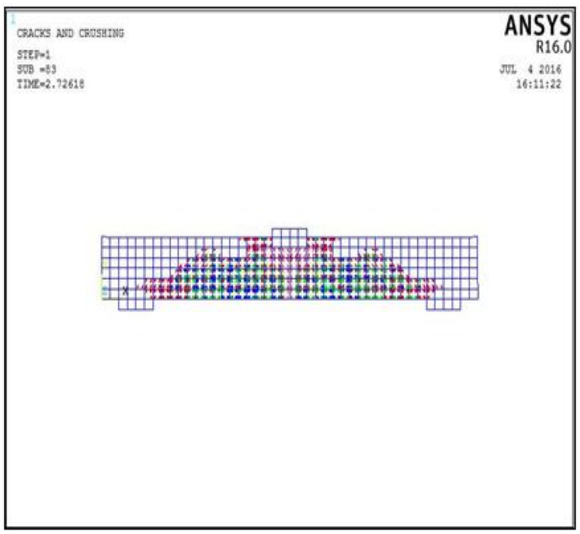
Cracks at 30 KN



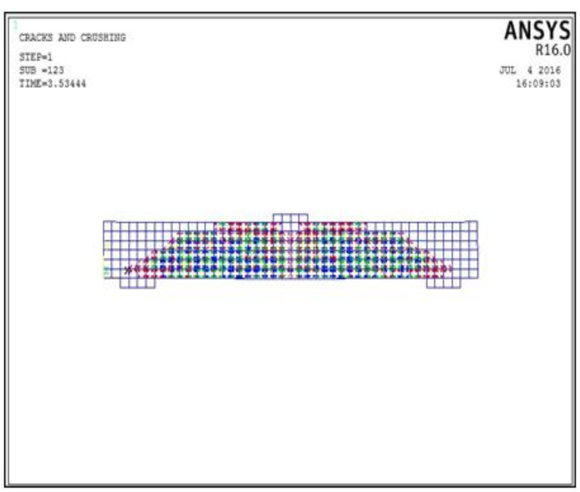
Deflection contour at 40 KN



Stress contour at 40 KN

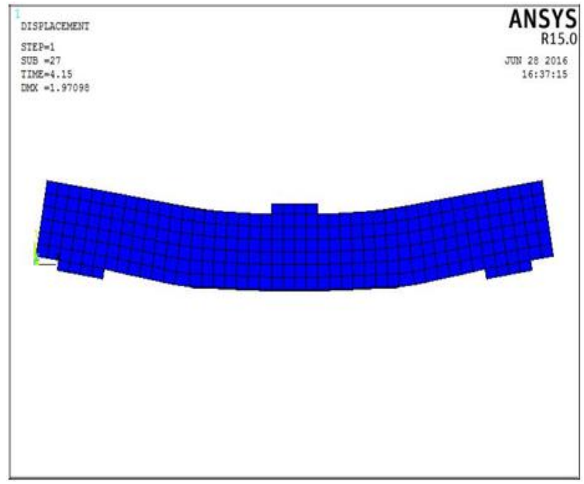


Crack at 40 KN

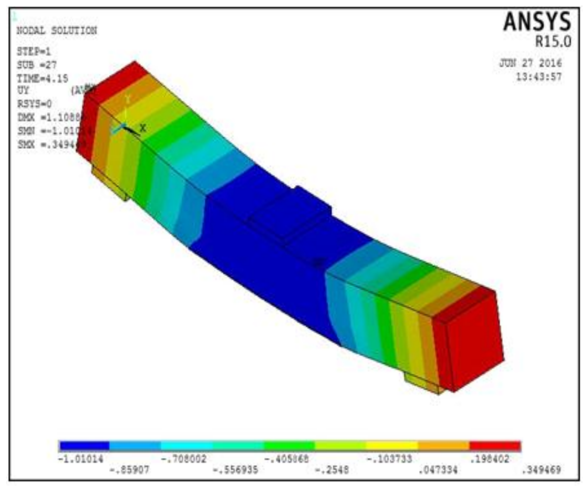


Crack pattern at ultimate load

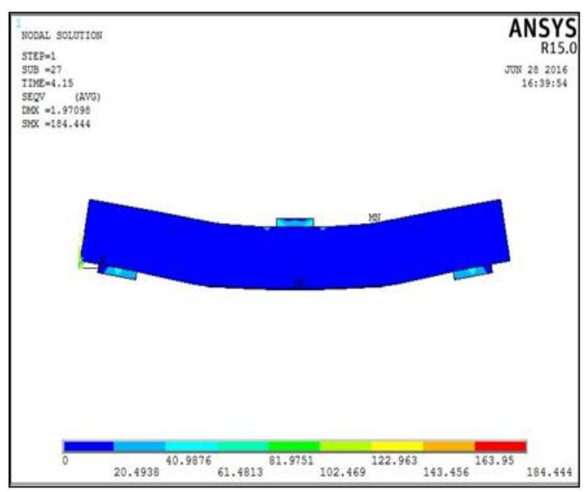
<u>450mm retrofit:</u>



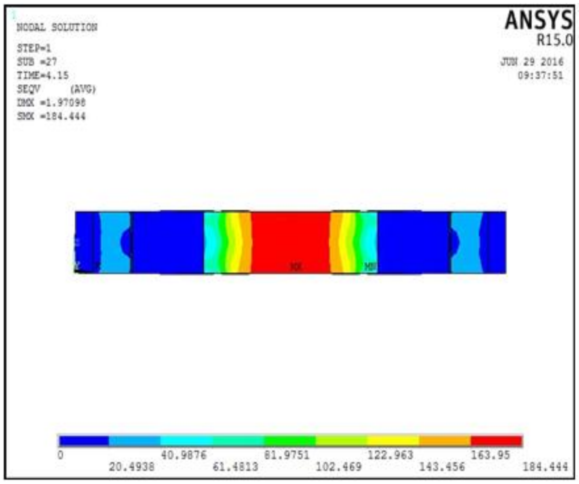
Deformation pattern at ultimate load



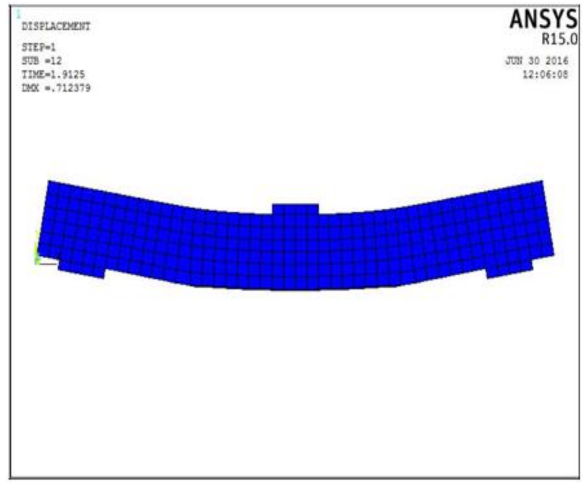
Deflection contour at ultimate load



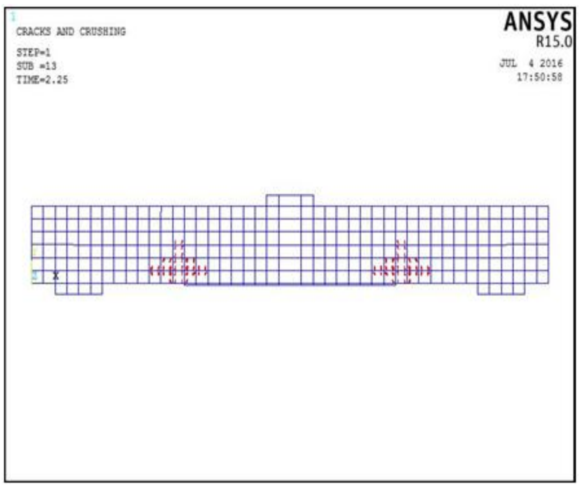
Stress contour at ultimate load



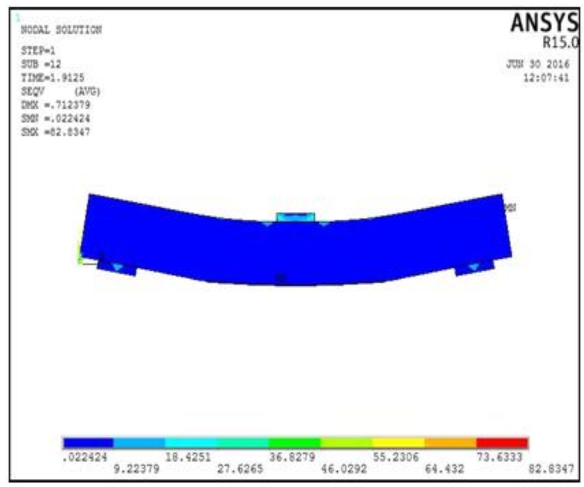
Stress contour at ultimate load bottom view



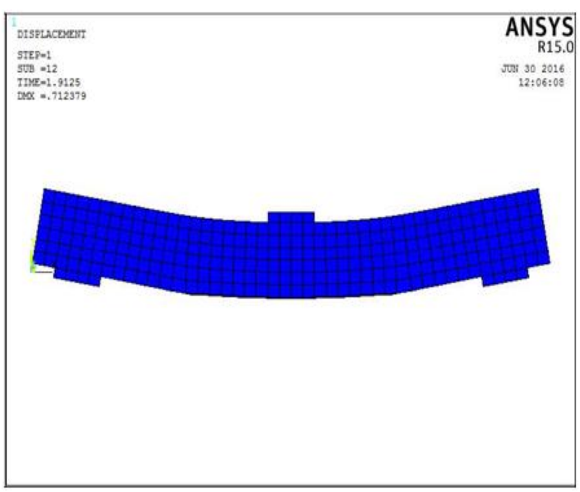
Deformed shape at 30 KN



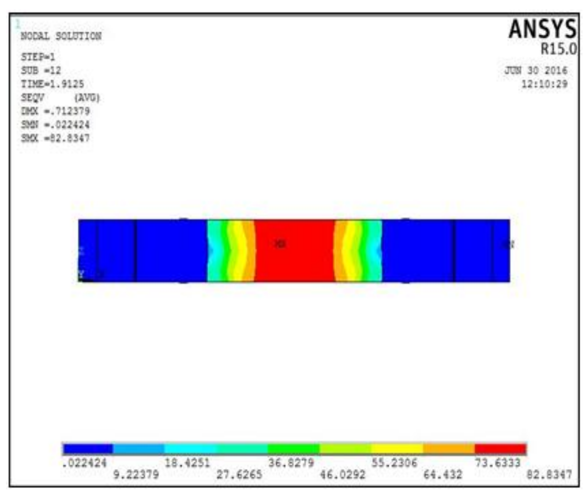
First crack



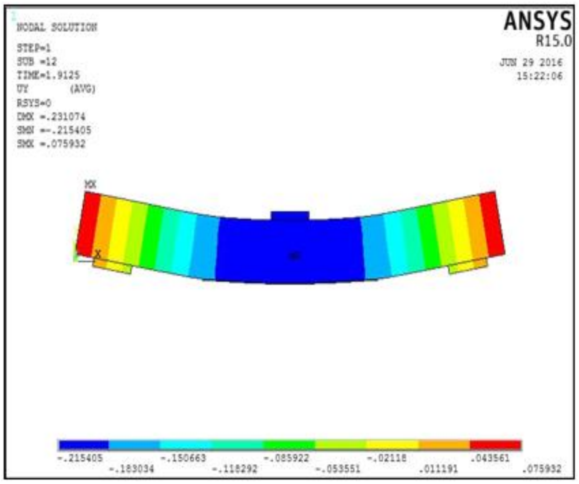
Stress contour at 30 KN



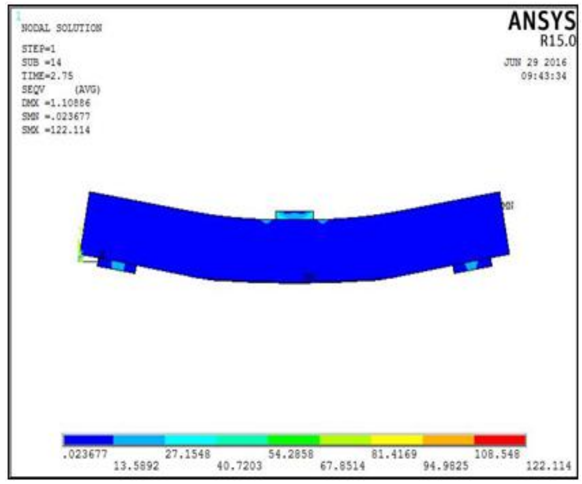
Deflection contour at first crack



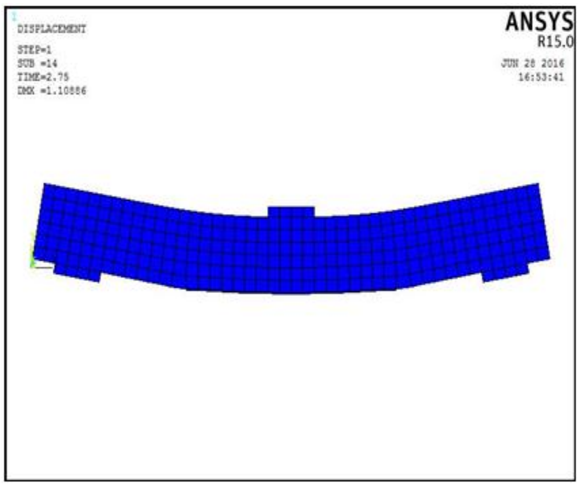
Stress contour at 30 KN bottom view



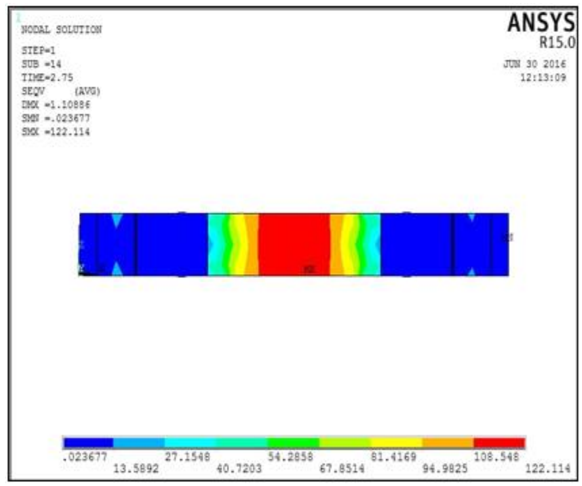
Deflection contour at 30 KN



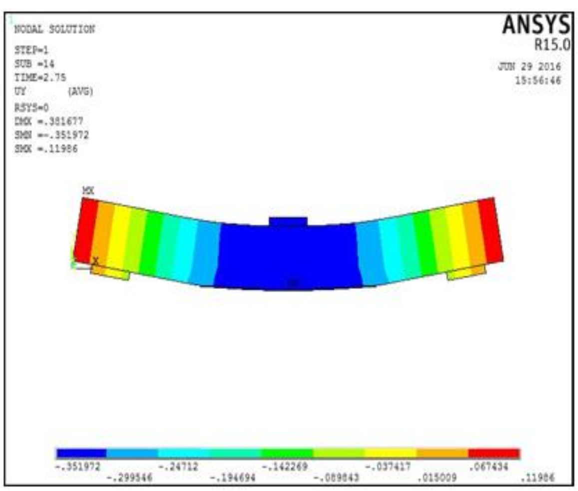
Stress at 40 KN



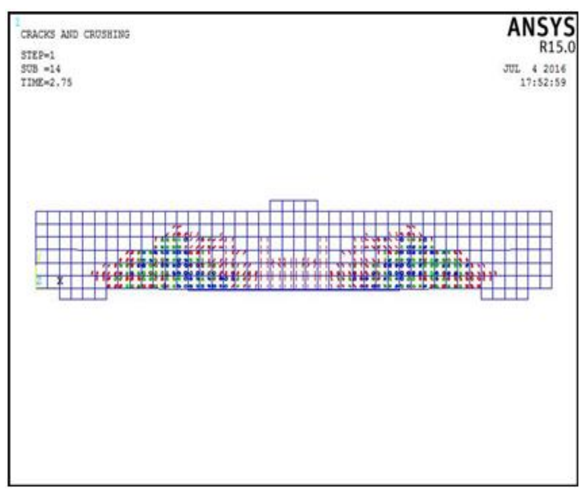
Deformed shape at 40 KN



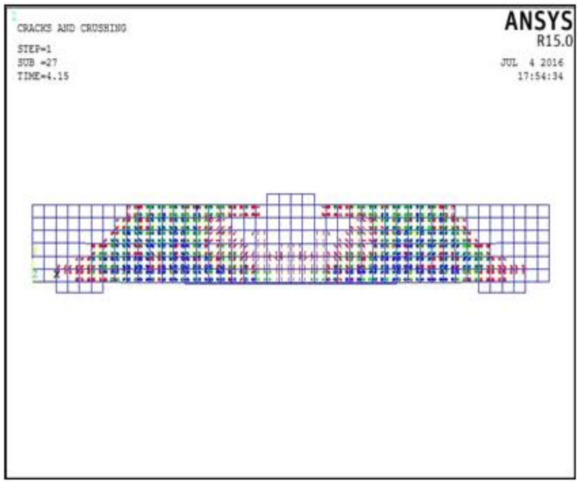
Stress contour bottom view at 40 KN



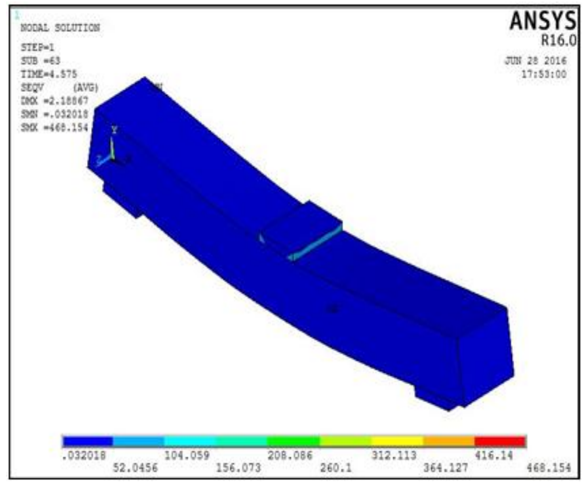
Deflection contour at 40 KN



Crack at 40 KN

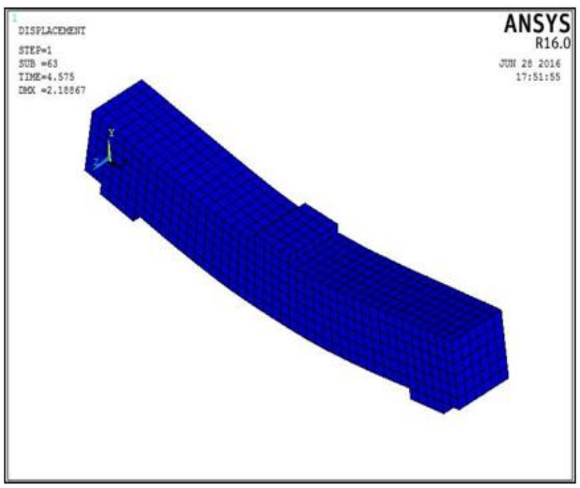


Crack at ultimate load

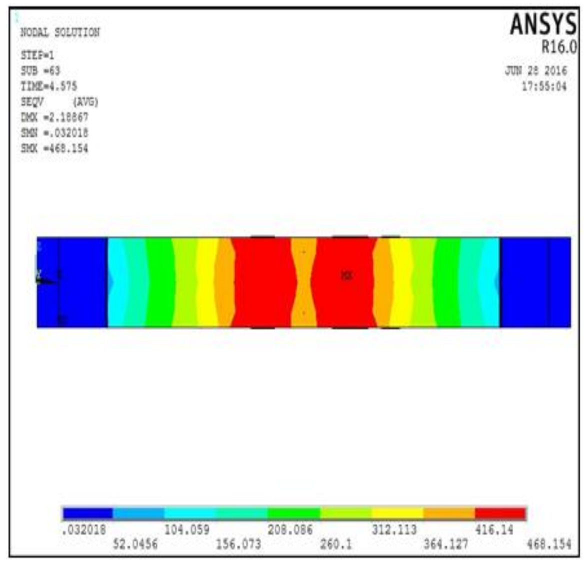


Stress contour at ultimate load

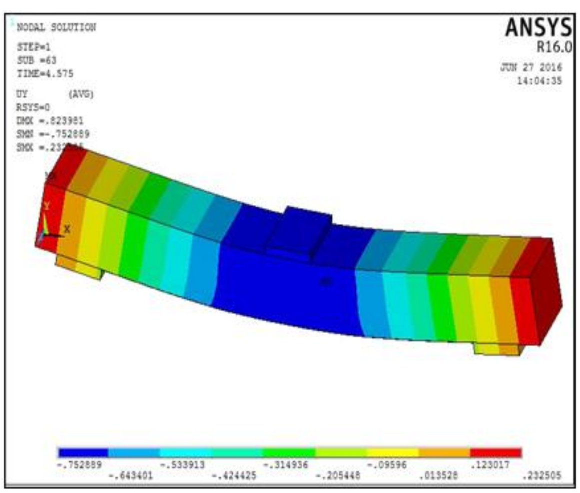
800 mm retrofit:



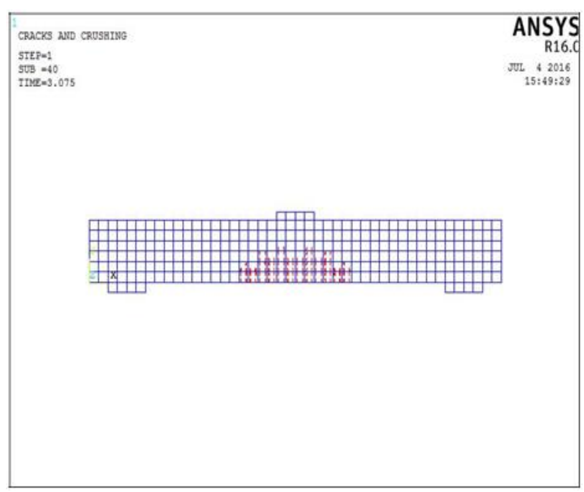
Deformation pattern at ultimate load



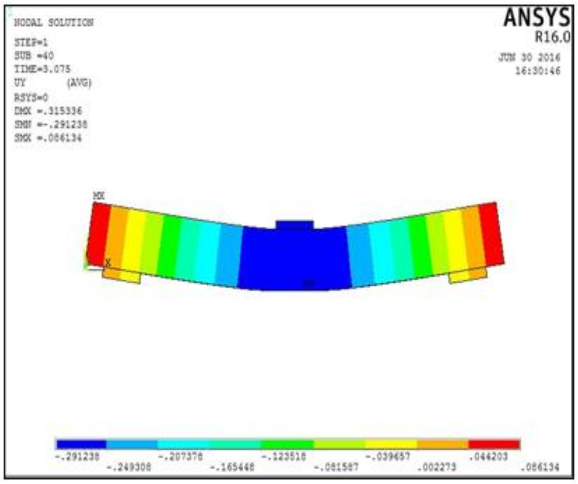
Stress contour at ultimate load bottom view



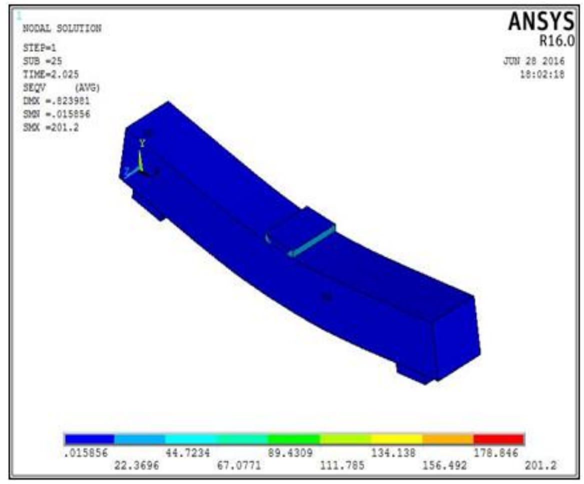
Deflection contour at ultimate load



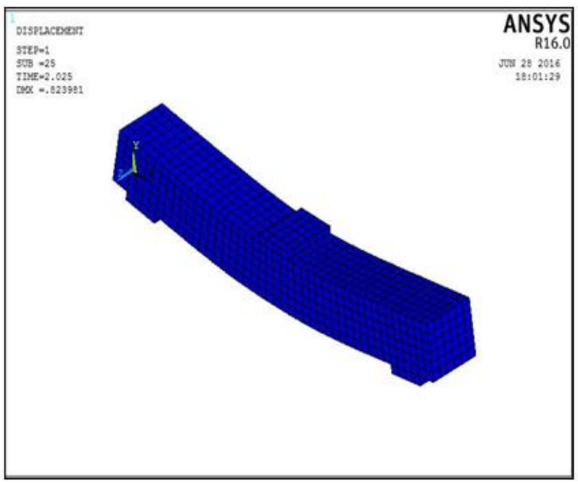
First crack



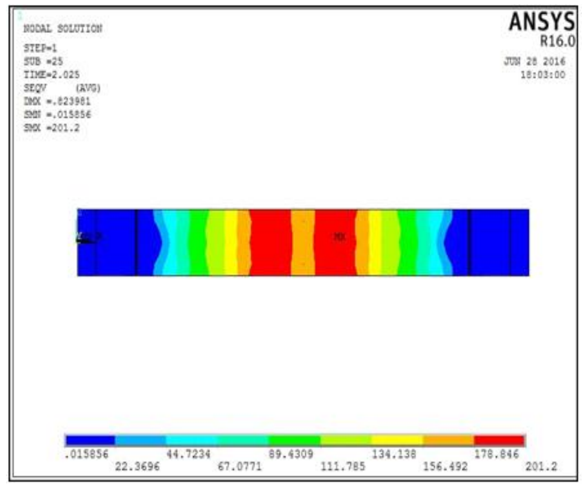
Deflection contour at first crack



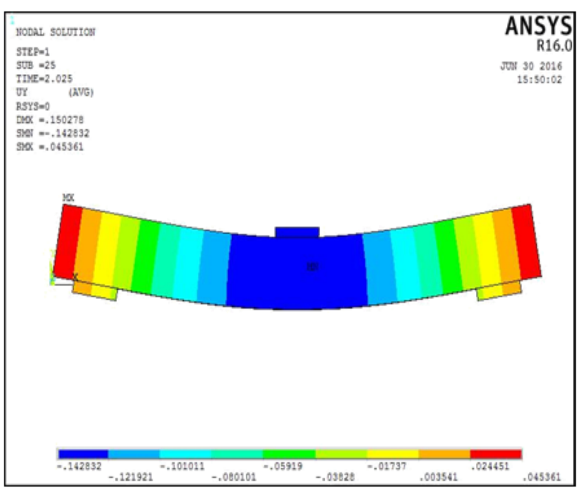
Stress contour at 30 KN



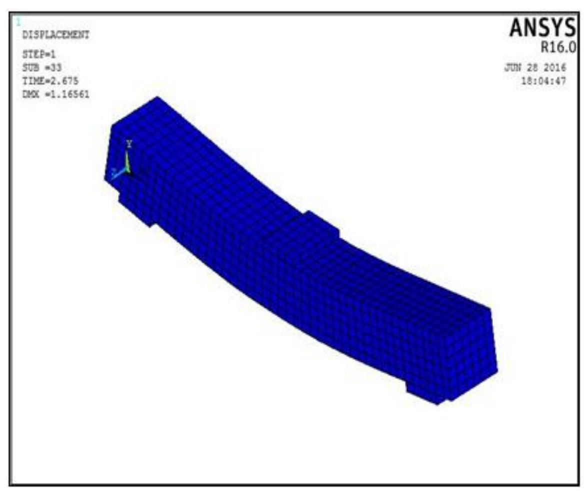
Deformation pattern at 30 KN



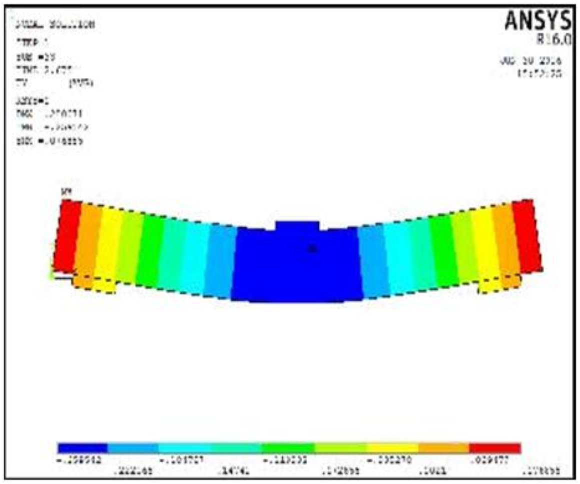
Stress contour at 30 KN bottom view



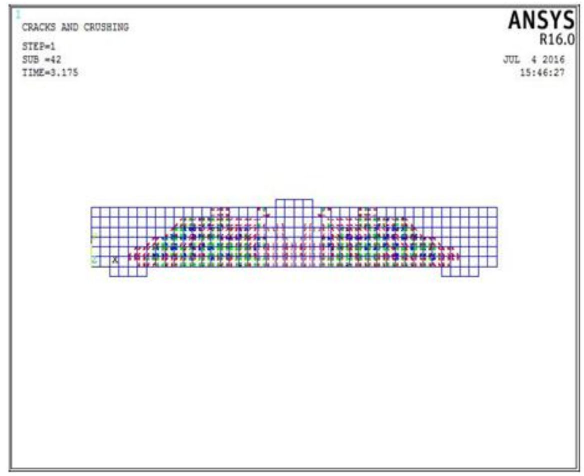
Deflection contour at 30 KN



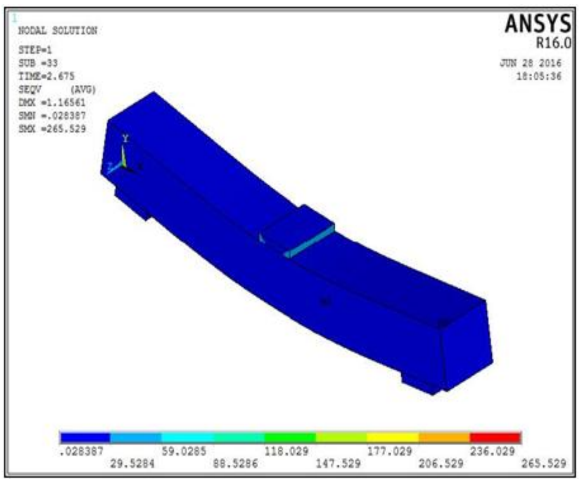
Deformation pattern at 40 KN



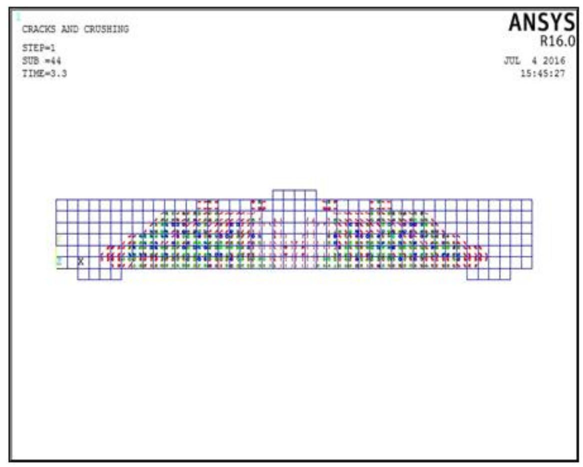
Deflection contour at 40 KN



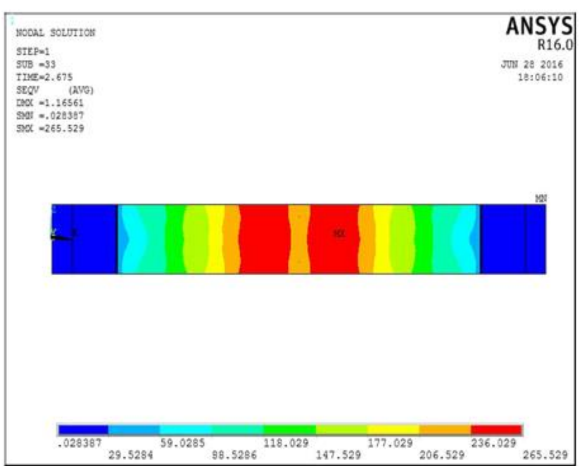
Crack at 47.62 KN



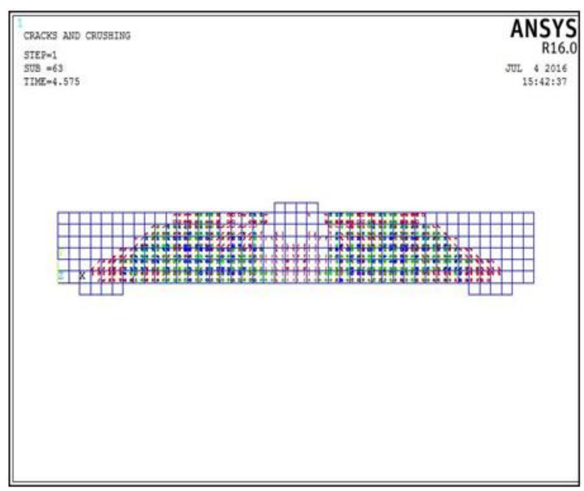
Stress at 40 KN



Crack at 49.5 KN



Stress at 40 KN bottom view



Crack pattern at ultimate load

The load-deflection values of Full Beam are tabulated below.

FULL BEAM

Load and deflection values for control beam

LOAD (KN)	DEFLECTION (mm)
1.6335	0.0188
5.26935	0.06
8.26935	0.095
12.081	0.139
13.2021	0.15
14.73	0.2
17.4	0.23
18.8235	0.27
20.292	0.29
22.455	0.31
28.305	0.334
30.1635	0.36
33.2865	0.387
35.2875	0.398
38.73	0.41
40.893	0.42
42.7695	0.45
44.307	0.52
49.137	0.71
53.016	1.15

Load and deflection values for 450 mm retrofitted beam

LOAD (KN)	DEFLECTION (mm)
1.5	0.015
5.25	0.052
8.55	0.085
13.65	0.116
16.6875	0.143
21.9375	0.165
25.3125	0.192
28.6875	0.21
33.75	0.25
41.25	0.35
47.25	0.512
52.5	0.634
57	0.85
60.75	0.998
62.25	1.1

Load and deflection values for 800 mm retrofitted beam

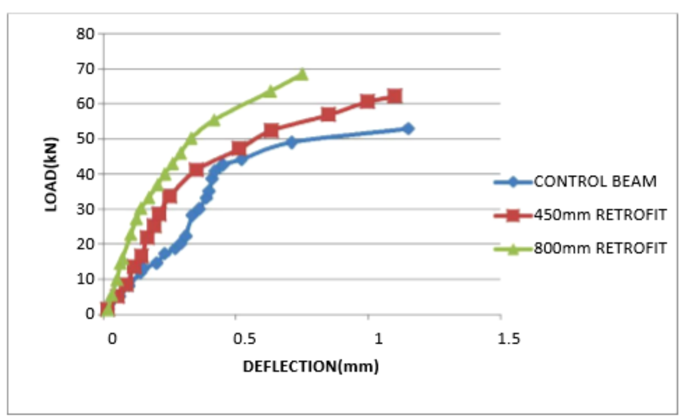
LOAD (KN)	DEFLECTION(mm)
1.5	0.016
5.625	0.03
10.125	0.05
14.625	0.063
16.125	0.071
22.875	0.102
27.375	0.124
30.375	0.14
33.375	0.17
37.125	0.203
40.125	0.23
43.125	0.26
46.125	0.29
50.25	0.33
55.5	0.417
63.75	0.63
68.625	0.75

		Control	450mm Retrofit	800 mm Retrofit
Full Beam	Ultimate Load(KN)	52.9	63.2	69
	Deflection at ultimate load (mm)	1.15	1.1	0.75
	First crack load (KN)	20.28	33.74	46.125
Experimental Results	Ultimate Load(KN)	53.2	60.85	-
	Deflection at ultimate load (mm)	2	1.8	-

Summary of the results

***Values shown in the brackets are the final results to be taken for the full beam**

The plot of Load versus Deflection for Full Beam models is presented in the figures below.



Load vs Deflection for full beam

VII. CONCLUSION

7.1 General

Finite Element Analysis to learn the feasibility of using externally attached steel plates to the tension face of reinforced concrete beams additional as external reinforcement has been presented. It involved the modelling of beams, out of which one was controlled beam and the rest were steel plate retrofitted beams. Two retrofitted models were studied one in which the entire span between the supporting has retrofitted (800 mm length), while the other retrofitted in the central half portion (450 mm length). Full-scale models of the beam were analysed.

Following resolutions were drawn out from the results obtained:

- ANSYS software is capable of predicting the ultimate load carrying capacity of reinforced concrete beam with great accuracy.
- For concrete, SOLID65 and steel reinforcement, LINK180 gives fairly good results
- The results obtained from ANSYS are consistent with that available in the literature.
- The ultimate failure load obtained in the case of control beam was in good agreement with the corresponding results in the literature with an error of 0.7 %.
- Full beams modelled taking considerations of the symmetry and results of failure load was observed.
- The ultimate strength of the retrofitted beams was found to be higher than the control beam. In the case of 450 mm retrofitted beam, the force increased by about 20% whereas in the case of 800 mm retrofitted beam, the increase in strength was found to be about 30%.
- From this, it can interfere the influence of length of retrofit is not a major factor in increasing the force whereas the position of the retrofitting material matters.
- Since the predominant mode of failure is flexure, the central portion of the beam needs to be retrofitted to achieve maximum strength.
- The load deflection plot is moderately comparable for full beam models

The load deflection plot is fairly for full beam models. Therefore symmetric conditions can be utilised for modelling in ANSYS.

7.2 Scope of Future Work

- In this research, discrete reinforcement approach has used. Smeared reinforcement approach may be used in ANSYS and validated.
- Analysis can be performed to study the behaviour of retrofitted beams by varying the number and width of the retrofitted plates
- This study can extend to other techniques of retrofitting
- Dynamic analysis may also be carried out

REFERENCES

[1] Ali Ahmed (2014). "Modelling of a reinforced concrete beam subjected to impact vibration using ABAQUS." *International Journal of Civil and Structural Engineering,* vol.4, ISSN 0976-4399.

[2] Ali Ghods, Mahyar Mir, Fahimeh Miri(2013). "A study of CFRP composites effects in seismic retrofitting of concrete connections using ABAQUS software." *International Research Journal of Applied and Basic Sciences,* Vol.6, ISSN 2251-838X.

[3] Alper Büyükkaragöz (2010)." Finite element analysis of the beam strengthened with prefabricated reinforced concrete plate."*Scientific Research and Essays, vol*. 5(6), ISSN 1992-2248 © 2010 Academic Journals, pp. 533-544.

[4] ANSYS Help Menu – ANSYS 15.0

[5] Anthony J Wolanski (2004). "Flexural Behaviour of Reinforced and Prestressed Beams Using Finite Element Analysis."*MS Thesis, Marquette University.*

[6] Balamuralikrishnan R and Antony Jeyasehar C (2009). "Flexural Behaviour of RC Beams Strengthened with Carbon Fibre Reinforced Polymer (CFRP) Fabrics."*The Open Civil Engineering Journal,* vol.3, ISSN 1874-1495, pp.102-109.

[7] Dhanu M.N, Revathy D, Lijna Rasheed and Shanavas S (2014). "Experimental and Numerical Study of Retrofitted RC Beams Using FRP."*International Journal of Engineering Research and General Science,* vol.2, Issue 3, ISSN 2091-2730.

[8] Farshid Jandaghi Alaee and Bhushan Lal Karihaloo (2003). "Retrofitting of Reinforced Concrete Beams with CARDIFRC." *Journal of Composites for Construction,* ASCE 1090-0268, pp. 174-186.

[9] Hee Sun Kim, Yeong Soo Shin (2011). "Flexural behaviour of reinforced concrete (RC) beams retrofitted with hybrid fiber reinforced polymers (FRPs) under sustaining loads." *Composite Structures* 93, ISSN 0263-8223, pp. 802-811.

[10] Jayajothi.P, Kumutha R and Vijai K (2013). "Finite element analysis of FRP strengthened RC beams using Ansys."*Asian Journal of Civil Engineering (BHRC),*vol. 14, no.4.

[11] Khairy Hassan Abdelkareem, Fayez Kaiser Abdelseed, and Mohamed Omar Sayed (2012)," Theoretical investigation on flexural performance of RC beams strengthened externally by CFRP." *Journal of Engineering Sciences, Assiut University,* vol. 40, No 1, pp.67-9.

[12] Namasivayam Aravind, Amiya K. Samanta, D. K. Singha Roy and Joseph V. Thanikal (2013). "Retrofitting of Reinforced Concrete Beams using Fibre Reinforced Polymer (FRP) Composites." *Journal of Urban and Environmental Engineering. v.7, n.1, p.164-175, ISSN 1982-3932*

[13] Omrane Benjeddou, Mongi Ben Ouezdou, Aouicha Bedday (2007). "Damaged RC beams repaired by bonding of CFRP laminates*." Construction and Building Materials* 21, pp.1301–1310.

[14] Sandeep kumar L.S, Dr.H.N.Jagannatha Reddy, Rumina Nizar (2013). "Retrofitting of RC Beams using Natural FRP Wrapping (NSFRP)." *International Journal of Emerging trends in Engineering and Development,* Issue 3, vol.5. ISSN 2249-6149

[15] T. Tejaswini and Dr.M.V.Rama Raju (2015). "Analysis of RCC Beams using ABAQUS*." International Journal of Innovations in Engineering and Technology*, vol.5, ISSN 2319-1058.

[16] Tej Sai M and Kantha Rao M(2014). "Retrofitting of Reinforced Concrete Beams using MS Steel Plates and Modelling using Finite Element Approach." *International Journal of Engineering Research & Technology,* Issue 9, Vol.3, ISSN 2278-0181.

[17] Umesh Basappa and Amritham Rajagopal (2013). "Modeling of CFRP strengthed RCC beam using the nonlinear finite element Method."*Journal of Structural Engineering,* vol.40, no.2, pp.169-184.

[18] Wael.B.Almajed and Robert Y. Xiao (2006). "Experimental Study of Retrofitted Flexural Reinforced Concrete Beams in Tension and Compression Areas with." *31st conference on our world in concrete and structures.* Article Online Id: 100031011

[19] Yasmeen Taleb Obaidat (2010). "Structural Retrofitting of Reinforced Concrete Beams using Carbon Fibre Reinforced Polymer." *Department of Construction Sciences, Structural Mechanics,* ISSN 0281-6679.

[20] Subramani.T, Athulya Sugathan, "Finite Element Analysis of Thin Walled- Shell Structures by ANSYS and LS-DYNA", International Journal of Modern Engineering Research,Vol.2, No.4, ISSN: 2249-6645, pp 1576-1587,2012