Available online at http://www.journalcra.com

INTERNATIONAL JOURNAL OF CURRENT RESEARCH

International Journal of Current Research
Vol. 13, Issue, 09, pp.18727-18732, September, 2021

DOI: https://doi.org/10.24941/ijcr.42045.09.2021

ISSN: 0975-833X

RESEARCH ARTICLE

FINITE ELEMENT ANALYSIS OF FLEXURAL BEHAVIOUR RECYCLED AGGREGATE CONCRETE BEAM USING ABAQUS

[1,*]Narayana Gowda, K.N., [2]Sadath Ali Khan Zai, [3]Chethankumar, J. and [4]Aswini Kumar K.N·

[1]Research Scholar, Dept of Civil Engineering, UVCE, Bangalore University, Bengaluru-560056
[2]Professor, Dept of Civil Engineering, UVCE, Bangalore University, Bengaluru-560056
[3]PG Student, Dept of Civil Engineering, UVCE, Bangalore University, Bengaluru-560056
[4]PG Student, Dept of Civil Engineering, Nagarjuna College of Engineering & Technology, Bengaluru Rural-562110

ARTICLE INFO

Article History:
Received 25th June, 2021
Received in revised form
20th July, 2021
Accepted 19th August, 2021
Published online 30th September, 2021

Key Words:
ABAQUS, Nonlinear Finite element analysis, Concrete Damage Plasticity, Recycled Coarse Aggregate, Treated Recycled Coarse Aggregate.

Corresponding author:
Narayana Gowda, K.N.,

ABSTRACT

This paper presents finite element analysis on flexural behavior of recycled aggregates concrete beam using Abacus and same is compare with the experimental obtained results., the numerical models for the static load appraisal on beam are starting to become more accurate and honorable, the ABAQUS program is used to model the behaviour of reinforced concrete (RC) beam subjected to static loading. The Nonlinear finite element model uses the concrete damaged plasticity (CDP) approach. The model used in the present study can help to compare the experimental investigations under considerations as a valuable data for the design aspect. Four test beam specimens of size 150X230X2200mm with four different mixes with partial replacement of Natural coarse aggregate with Recycled coarse aggregate (RCA) and Treated recycled coarse aggregate (TRCA) are modelled in ABAQUS 6.14 and tested for static loading and the parameters such as first crack load, ultimate load and load deflection behavior were studied, then the analytical results are compared with experimental results. Hence, the Nonlinear finite element models can simulate the behavior of recycled aggregate beam under static load and there was good compromise of analytical results obtained from ABAQUS 6.14 with that of experimental results.

Citation: Narayana Gowda, K.N., Sadath Ali Khan Zai, Chethankumar, J. and Aswini Kumar K.N. "Finite element analysis of flexural behaviour recycled aggregate concrete beam using abaqus*", 2021. International Journal of Current Research, 13, (09), 18727-18732.*

INTRODUCTION

The deposition of construction garbage which is increasingly accumulated due to various causes such as demolition of old construction is also an environmental concern. In India, the Central Pollution Control Board has assessed that the solid waste generation is about 48 million tonnes per annum of which 25% are from the construction industry. This scenario is not so different in the rest of the world. In order to decrease the construction waste, recycling of waste concrete as aggregate is beneficial and effective for preservation of natural resources. In order to eliminate the problem of waste is by reusing, recycling and reducing the construction materials in activities of construction. The use of recycled coarse aggregates (RCA) in structural concrete reduces the aggregates transportation distances, CO_2 emissions, and landfill space required for the construction waste material by enhancing the protection of the natural environment in a sustainable way.

The Finite Element Analysis (FEA) is the simulation of any given physical phenomenon using the numerical technique called Finite Element Method (FEM). Engineers use FEA software to reduce the number of physical prototypes and experiments and optimize components in their design phase to develop better products, faster while saving on expenses. The use of FEA tools has become widespread due to increased computation power and the ability of FEA software packages to simulate incredibly complicated components, structures and systems under a wide variety of situations and loading conditions.

Very few analytical investigations have been conducted under static loading for concrete slab that contain combination of untreated and treated recycled coarse aggregates. So, in this study a Finite element analysis is carried out using ABAQUS 6.14.

LITERATURE REVIEW

The present analytical analysis is concentrated on static flexural behaviour of slab using RCA and treated RCA as partial replacement for NCA concrete. Numerous researches focused their studies on Recycled aggregate concrete and analytical investigation using Finite element method of Reinforced Concrete components. The Properties of recycled concrete aggregate under different curing conditions (air, water, paint) and increasing the concrete age led to an increase in its ultimate strength. According to the Madan Mohan Reddy K et al**,** investigated the use of construction and demolition waste as a Recycled Concrete Aggregate (RCA) in the production of new concrete. The studies were conducted with an M20 mix with the selected w/c ratio:0.5 and the development of compressive strength of the RAC and NAC at the age of7 & 28 days were assessed. The result shows the compressive strength of RAC is on average 87% of the NCA and the Slump of RAC is low and that can be improved by using Saturated Surface Dry condition of RCA [2,3] Conducted an experimental study on recycled aggregate concrete obtained from the construction demolition waste and used it as a full replacement for natural coarse aggregate with mix ratio 1:2:4 and w/c of 0.5, and carried out compressive strength test at 7,14,21, 28 days. The 28 days compressive strength of 12 batches of recycled aggregate concrete which they have obtained is 29.42 N/mm^2 which were greater than characteristic compressive strength of 25 N/mm^2. . The experimental and the theoretical studies about the elasticity modulus and energy capacities [4]. Fifteen concrete mixtures are produced with different ratios of replacement of natural coarse aggregate with recycled coarse aggregate, and at the age of 28 days, the tests (compressive strength, modulus of elasticity in compression, splitting tensile strength and density) are applied on the specimens (150mm diameter and 300mm length cylinder). [5]A 3D finite element model was developed to simulate the performance and various failures modes of plain concrete in comparison with geopolymer concrete by using Abaqus software. In this modeling, a continuum stress/displacement three-dimensional element eight node brick reduced integration (C3D8R) element of plain concrete beam having dimensions of 500 X 100 X 100 mm is modeled. The analytically determined values were found to be in relation with the experimentally obtained values. the deformation pattern of the both concrete specimens were found to be same. [6] The numerical simulation of a reinforced concrete structure requires an accurate model of the structural elements and its constituent members acting as a composite made up of concrete and steel. A sketch of each section is created separately with ABAQUS, which can then be extruded in any direction; this is why a 3D solid element in “modeling space” using deformable type for beam was created. In order to develop concrete beam, 8- node continuum solid element was utilized. 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. Necessary partitions of the concrete beam (of size 1200 x200 x100) are made to facilitate load application and meshing. The steel reinforcement of size 1150 mm is modeled as two –node beam elements connected to the nodes of adjacent solid elements. Deflections and stresses at the centerline along with initial and progressive cracking of the finite element model compare well to experimental data obtained from a reinforced concrete beam. [7] Development of high strength concrete as a new ecological construction material to sustain the gradually expanding construction industry has arisen. This paper presents nonlinear finite element analysis of three-dimensional high strength reinforced concrete beams using ABAQUS. The uniaxial compressive strength for the beam models were taken from the existing experimental data on high strength concrete cubes. Euro code 2 was also used to establish material parameters for the models for concrete and reinforcing bars. In this study, two150mm x 200mm x 1200mm simply supported rectangular concrete beam models subjected to loads at different shear span to effective depth ratios (a/d = 1.0 and 2.0) were analysed. Numerical results were validated with the existing experimental data specifically on the load-deflection responses von mises stresses. It was found that the finite element results show greater than 70% agreement with the experimental results. used the ABAQUS program to model the behaviour of reinforced concrete (RC) beams. The size of beam used was 305 X 770 X 6095mm and was modelled in ABAQUS using model proposed by Saenz [8]. The results indicated that the displacement, tensile strain for the main reinforcement, compressive strain for concrete and crack patterns obtained from the finite element model (FEM) are well matched with the experimental results. A 3D model of a concrete cube was prepared using smeared crack model and concrete damage plasticity approach by [9]. A comparative study for cube of size 150mm modelled in ABAQUS using C3D8 element was done using different models and their results were discussed. [10] presented a finite element analysis of eight identical beams of size 150 X 300 X 1960mm for the effect of retrofitting with Carbon fiber reinforced polymer. A nonlinear finite element analysis was carried out using stress strain relationship proposed by Saenz[11]. Hence an attempt has been made in the present study to validate existing Finite element model with [8] in ABAQUS 6.14 and the same concrete damaged plasticity model is used to compare the experimental results obtained.

EXPERIMENTAL WORK

As from the experimental results obtained for reinforced aggregate concrete was prepared by partial replacement of crushed concrete coarse aggregates with natural coarse aggregates. The study also includes treating the recycled concrete aggregates with Nitoflor Lithurin (basically mixture of sodium silicate and lithium silicate solutions) to enhance the surface properties of RCA. Twelve identical beam of size 150X230X2200mm as shown in Figure 1 with four different mixes (i) B-1:100%NCA(conventional concrete of grade M-25) (ii) B-2:50%TRCA+50%NCA (M-25 grade concrete made using 50% treated recycled concrete aggregate and 50% natural coarse aggregate)(iii) B-3:50%TRCA+30% NCA+20% RCA,(M-25grade concrete made using 50% treated recycled concrete aggregate, 30% natural coarse aggregate and 20% recycled concrete aggregate) and(iv) B-4: 80%NCA+20%RCA (M-25 grade concrete made 80% natural coarse aggregate and 20% recycled concrete aggregate) were casted and tested for static loading behaviour at Department of Civil Engineering, UVCE, Bangalore University, Bengaluru, and Karnataka, India. load-deflection curve for all the four beam is presented in the Figure 2. The results like as first crack load, ultimate load and load deflection behavior, ductility index, energy absorption capacity, toughness index, strain analysis cracking moment and moment of resistance are determined and also the theoretical and experimental load were compared.

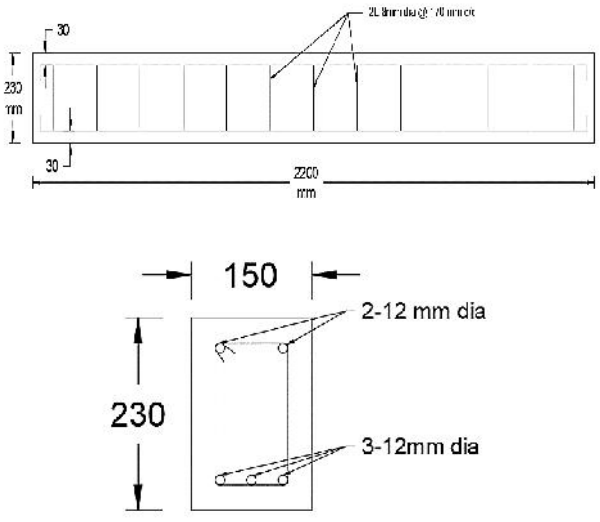

Fig. 1. Geometry and reinforcement arrangement of beams

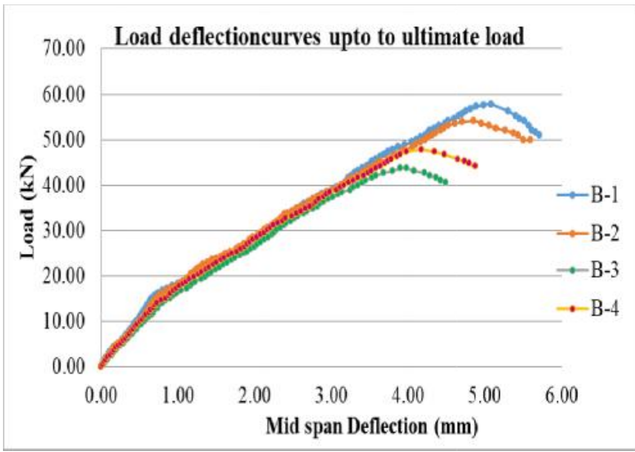

Fig. 2. Experimental load-deflection curve of all specimens

FINITE ELEMENT ANALYSIS

Finite element analysis was performed to model the nonlinear behaviour of the recycled aggregate beam of size 150X230X2200mm for four different matrices (i) B-1 (conventional concrete 100% NCA), (ii)B-2 (50% TRCA +50% NCA containing concrete) (iii)B-3 (50% TRCA+30% NCA+20% RCA containing concrete) and (iv) B-4 (80% NCA +20% RCA containing concrete). The FEA package ABAQUS 6.14 was used for the analytical analysis of static load behaviour of recycled aggregate beam.

Material Properties and constitutive Models used

Concrete: Concrete is defined as an isotropic material before yielding and cracking model is defined for nonlinear analysis. 3D solid element is used for modelling of concrete as C3D8R which indicates eight nodes in three degrees of freedom in x,y and z directions. The compressive strength obtained from experimental studies f_{ck} is 31.7N/mm^2. The linear properties used in ABAQUS 6.14 for the analytical analysis are shown in Table 1. To predict the behaviour of concrete, non-linear analysis using Concrete damage plasticity model because it uses concepts of isotropic damage elasticity in conjunction with isotropic tensile and compressive plasticity to represent the inelastic behaviour of concrete. The parameters for nonlinear behaviour of concrete used in ABAQUS 6.14 are

Table 1. Linear Properties of Concrete

Parameter	Value
Concrete Density	2500 kg/m^3
Young's Modulus	5000X $\sqrt{f_{ck}}$=28,151.37N/mm^2
Poisson's ratio	0.17

Table 2. Nonlinear properties of concrete

Parameter	Value
Dilation angel (φ)	35°
Plastic potential eccentricity (e)	0.1
Initial biaxial/uniaxial ratio (f_{bo} / f_{co})	1.16
Shape of the loading surface (K_c)	0.66
Viscosity parameters	0

shown below. The stress strain relationship for compressive behaviour of concrete is derived from the relationship proposed using stress strain in uni axial compression by Saenz [11]

$$\sigma_c = \frac{E_c \times \epsilon_c}{1 + (R + R_E - 2)\left(\frac{\epsilon_c}{\epsilon_0}\right) - (2R - 1)\left(\frac{\epsilon_c}{\epsilon_0}\right)^2 + R\left(\frac{\epsilon_c}{\epsilon_0}\right)^3}$$

Where

$$R = \frac{R_E(R_\sigma - 1)}{(R_\epsilon - 1)^2} - \frac{1}{R_\epsilon},$$

$$R_E = \frac{E_c}{E_o}$$

$$R_E = \frac{E_c}{E_o}$$

σ_c = Stress in concrete
ϵ_c = Strain in concrete
E_c = Initial modulus of elasticity
f'_c = maximum compressive strength of concrete.
Strain ratio; $R_\epsilon = \epsilon_f / \epsilon_0 = 4$
Stress ratio; $R_\sigma = f'_c / \sigma_f = 4$
ϵ_f and σ_f are maximum strain and corresponding stress on the uniaxial stress-strain curve
ϵ_0 = Strain corresponding to f'_c in a uniaxial compressive test = 0.0025.
Secant Modulus: $E_0 = f'_c / \epsilon_0$

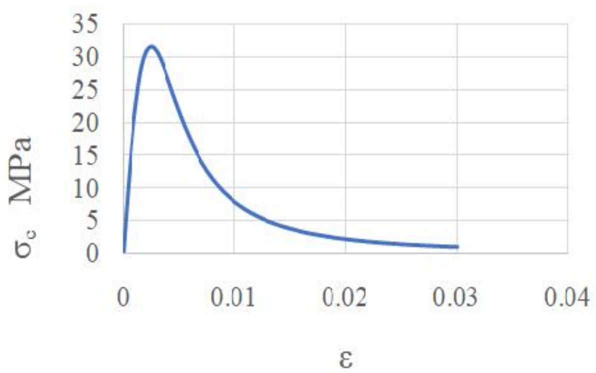

Fig. 3. Stress-strain relationship of concrete under uniaxial compression by Saenz [11]

The tensile strength of concrete f_{cr} is calculated as per IS 456:2000[15] The tensile behaviour of concrete is obtained by the stress strain relation under uni axial tension proposed by Massicotte[12] as shown in Figure

$f_{cr} = 0.7 \text{ X } \sqrt{f_{ck}} = 3.94$ N/mm^2

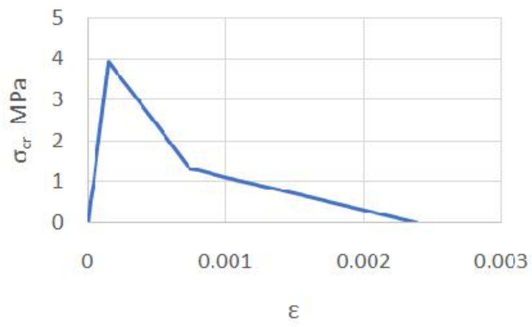

Fig 4. Stress-strain relationship of concrete under uniaxial tension by Massicotte[12]

Steel: Steel reinforcement was modelled as 3D truss element as T3D2 having 2 node displacement and assumed to be elastic perfectly plastic material. The reinforcement steel used in the present investigation is HYSD bars having yield strength f_y = 500 N/mm^2 and elastic modulus E_s = 200 GPa. A poisson's ratio of 0.3 was used for steel reinforcement. The model is assembled using translation and rotation options in ABAQUS 6.14 according to their geometry as shown in Figure 5.

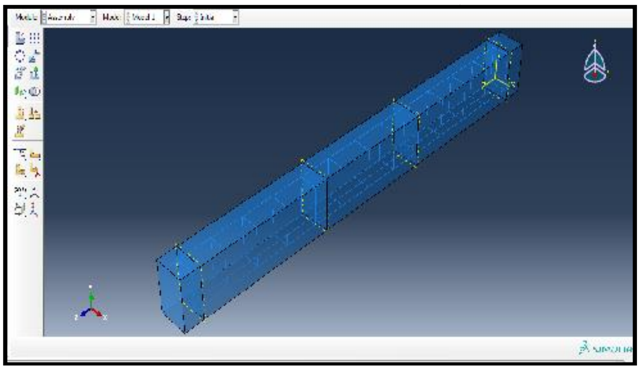

Fig. 5. Modelling of beam in ABAQUS 6.14

Numerical Analysis: The interface between steel reinforcement and concrete was considered as perfect bond. The steel reinforcement was embedded into the host region of concrete. Meshing is important in the FEM analysis which comprise of shape and size of element. The meshing of model was done before loading condition and was discretized into finite elements. For solid element rectangular mesh of size 40mm was used as shown in Figure 6.

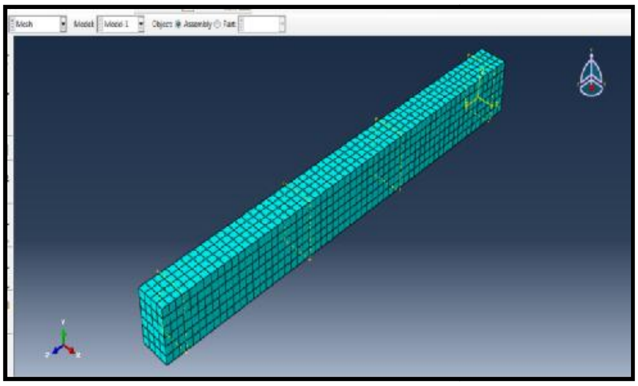

Fig. 6. Meshing of beam in ABAQUS 6.14

The bottom of the beam was simply supported on both ends. A load of 58kN is applied on top of the beam, i e 28kN on both side as it was two-point loading as shown Figure 7.

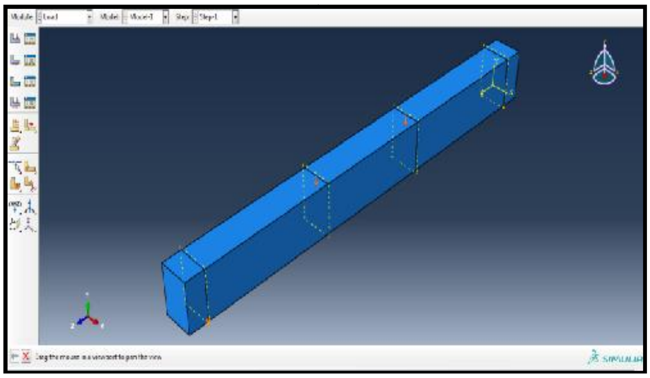

Fig. 7. Loading and Boundary Conditions of beam in ABAQUS 6.14

RESULTS

Analytical results of static loading on recycled aggregate concrete beam using ABAQUS 6.14 for specimens (i) B-1 (conventional concrete 100% NCA), (ii)B-2 (50% TRCA +50% NCA containing concrete) (iii)B-3 (50% TRCA+30% NCA+20% RCA containing concrete) and (iv) B-4 (80% NCA +20% RCA containing concrete) are shown below.

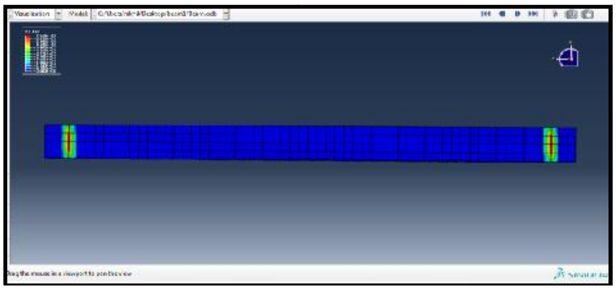

Fig. 8. Reaction Force B-1: 100% NCA in ABAQUS 6.14

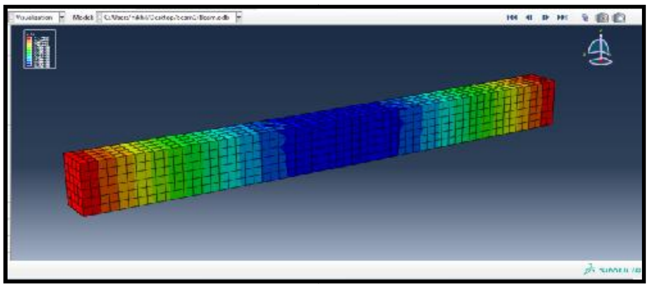

Fig. 9. Deflection of Beam B-1:100% NCA in ABAQUS 6.14

The analysis is run for the requisite parameters and models and the results are visualized in the ABAQUS 6.14. The obtained output results are, Reaction force shown in Figure 8 and deflection at mid-point of beam shown in Figure 9. From observing load versus s deflection curves from ABAQUS 6.14, the following data is obtained in the present investigation for beam specimens of size 150 X 230X 2200mm subjected to static bending load and the following results were investigated.

- Deflection Behavior
- First Crack Load
- Ultimate Load

Deflection Behavior: The Deflection Behavior is obtained from load-deflection curve after static loading analysis carried out in ABAQUS 6.14 are shown forB-1, B-2, B-3and B-4 below.

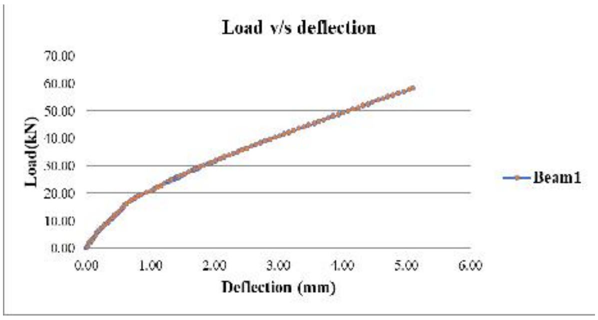

Fig 10. Load-deflection curve for beam B-1(100% NCA) in ABAQUS 6.14

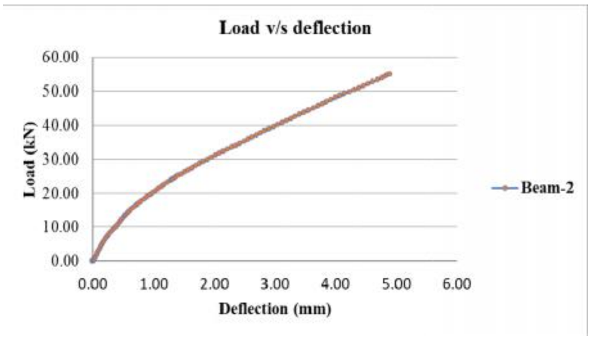

Fig 11. Load-deflection curve for beam B-2(50%trca+50% NCA) in ABAQUS 6.14

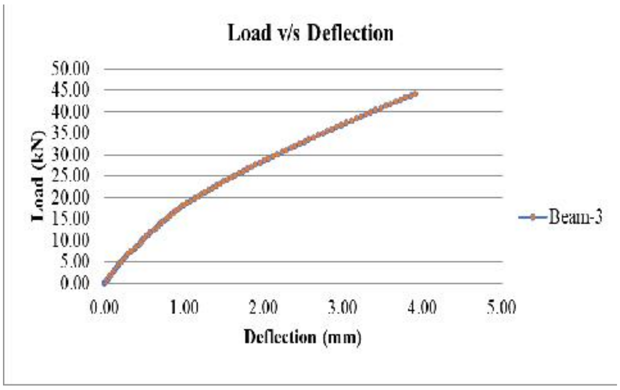

Fig 12. Load-deflection curve for beam B-3(50% TRCA+30% NCA+20% RCA) in ABAQUS 6.14

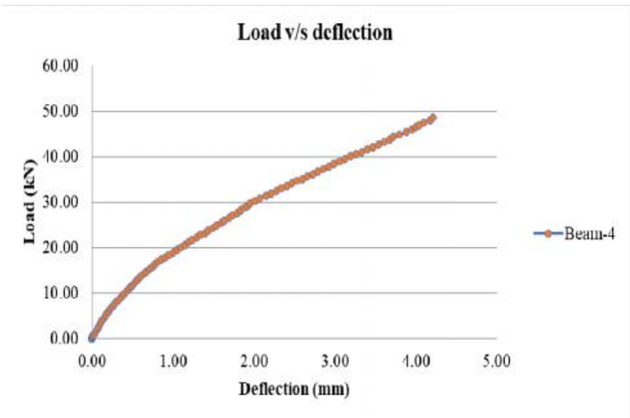

Fig 13. Load-deflection curve for beam B-4(80% NCA +20% RCA containing concrete)) in ABAQUS 6.14.

First crack Load: The first crack load is obtained from load-deflection curve after static loading analysis carried out in ABAQUS 6.14 are shown below.

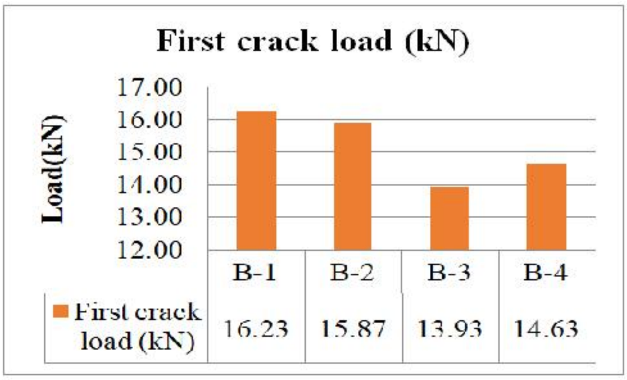

Fig 14. First crack load

Ultimate load: The ultimate or failure load obtained for four different mixes are shown are shown below. The analytical results obtained from ABAQUS 6.14 are compared with respect to the experimental results are as follows

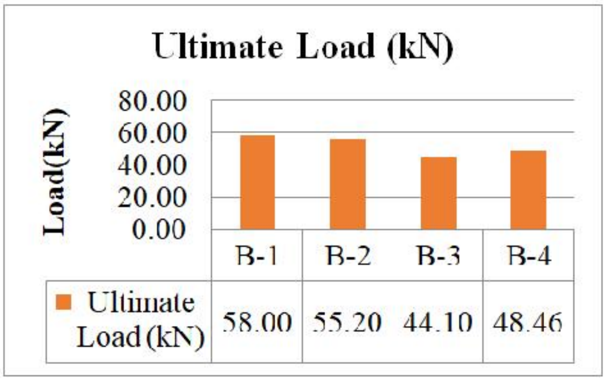

Fig 15. Ultimate-deflection

Comparison of Experimental and Analytical load deflection curve: The analytical and experimental load deflection curve are compared

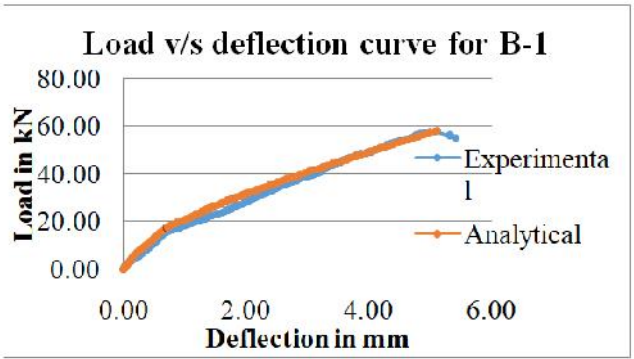

Fig. 20. Comparison of Experimental and Analytical values of beam B-1 load deflection curve

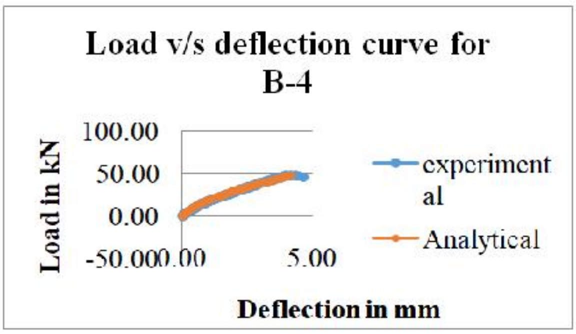

Fig. 21. Comparison of Experimental and Analytical values of beam B-4 load deflection curve

Comparison of Experimental and Analytical load (ABAQUS 6.14) First crack load: The analytical First crack load and experimental first crack load are compared and the ratio of experimental to analytical results (E/A) are shown below

Table 3. Comparison of First crack load

Designations	Experimental first crack load (kN) (E)	Analytical first crack load (kN) (A)	Ratio (E/A)
B-1	15.10	16.23	0.93
B-2	14.80	15.87	0.92
B-3	13.54	13.93	0.97
B-4	14.06	14.63	0.96

Comparison of experimental and analytical (ABAQUS 6.14) Ultimate load: The analytical Ultimate load and experimental Ultimate load are compared and the ratio of experimental to analytical results (E/A) are shown below.

Table 4. Comparison of Ultimate load

Designations	Experimental ultimate load (kN) (E)	Analytical ultimate load (kN) (A)	Ratio (E/A)
B-1	57.81	58	0.99
B-2	54.15	55.2	0.98
B-3	43.75	44.10	0.99
B-4	47.92	48.46	0.98

CONCLUSION

The load-deflection curve beyond peak load and up to the descending post-peak was recorded till the load reduced to 80% ~ 95% of the peak load (Pu). It is observed that all curves show linear variation up to the first crack load and exhibit non-linearity with further increase in load. The experimentally and analytically obtained values first crack load for B-1 is 15.10 whereas for B-2, B-3, B-4 are 14.80, 13.54, 14.06 and 16.23, 15.87, 13.93, 14.63 respectively. Percentage variation of experimentally and analytically obtained values first crack load for B-1, B-2, B-3 and B-4 are6.9%, 6.7%,2.7%,3.8 respectively. Both experimental and analytical analysis is compared and it was observed that the analytical first crack loads are good in agreement with the experimental first crack load of the recycled aggregate beam. The experimentally and analytically obtained values ultimate load for B-1 is 57.81 whereas for B-2, B-3, B-4 are 54.15, 43.75, 47.92 and 58.00, 55.20, 44.10, 48.46 respectively. Percentage variation of experimentally and analytically obtained values for ultimate load B-1, B-2, B-3and B-4 are 0.32%,1.9%, 0.79%, 1.1% respectively. Experimental and analytical values are compared and it can be seen that the analytical ultimate deflection matches the experimental ultimate deflection of the recycled aggregate Beam.

BIBILOGRAPHY

Narayana Gowda K.N & Dr. Sadath Ali Khan Zai (2021) "Experimental Investigation on Recycled Aggregates for Sustainable Concrete On beam Specimens Under static Loadings" International Research Journal of Engineering and Technology. Volume: 08 Issue: 02

Ahmed shaban Abdl-Hay (2017), "Properties of recycled concrete aggregate under different curing condition", Housing and Building National Research Centre, Volume 13

Hyginus E. Opera (2016), "Experimental study of concrete using recycled coarse aggregate", International Journal of Materials and Structural Integrity, Vol. 10, No. 4

H. Dilbas, O ¨. Cakir & M. Simsek "Recycled Aggregate Concretes (RACs) for Structural Use: An Evaluation on Elasticity Modulus and Energy Capacities", Int J Civ Eng.

Chithambar Ganesh," Finite Element Analysis over Geopolymer Concrete using Abaqus".

T. Tejaswini, "Analysis of RCC Beams using ABAQUS", PG Student, Dept. of Civil Engineering CBIT, Hyderabad, Telangana, India.

Oh, CL. "Finite element analysis of high strength reinforced concrete beams.", Faculty of Civil Engineering, University Teknologi MARA Malaysia,40450 Shah Alam, Selangor, Malaysia

Ali Raza, Qaiser uz Zaman Khan, and Afaq Ahmad, 2019. "Numerical Investigation of Load-Carrying Capacity of GFRP-Reinforced Rectangular Concrete Members Using CDP Model in ABAQUS". Advances in Civil Engineering Volume 2019

Tao Y. and J. F. Chen, M. ASCE 2015. "Concrete Damage Plasticity Model for Modeling FRP-to-Concrete Bond Behaviour". American Society of Civil Engineers, J. Compos. Constr. 2015.

Aydin Demir, Hakan Ozturk and Gokhan Dok (2016), "3D Numerical Modeling of RC Deep Beam Behaviour by Nonlinear Finite Element Analysis". DISASTER SCIENCE AND ENGINEERING

Saenz LP, Discussion of Paper By Desai P, Krishnan S. "Equation for stress-strain curve of concrete", Journal of ACI, Proc., No. 9, 61(1964) 1229-35B.

Massicotte, A.E. Elwi, J.G. MacGregor, Tension stiffening model for planar reinforced concrete members, ASCE Journal of Structural Engineering 116 (11) (1990).

IS 456:2000 Plain and reinforced code of practice

****