Novelty Journals

ISSN 2394-7357

International Journal of Novel Research in Civil Structural and Earth Sciences
Vol. 6, Issue 3, pp: (14-28), Month: September - December 2019, Available at: www.noveltyjournals.com

INVESTIGATION ON SPIRAL STIRRUPS IN REINFORCED CONCRETE BEAMS

KARUNANIDHI.S

ASST PROFESSOR, CIVIL ENGINEERING DEPT, COLLEGE OF ENGG&TECH, WOLLEGA UNIVERSITY, POST BOX NO:395,NEKEMTE,ETHIOPIA.

karunabalasingam@gmail.com

Abstract: **In recent times, the high cost and general shortage of reinforcing steel in many parts of the world has led to increasing interest in the possible use of alternative locally available materials for the reinforcement of concrete. Also, heavy loads on concrete beams produce diagonal shear cracks. Cracking in beams is normal and indicates the tension bars are actually working. Excessive cracking needs to be controlled by additional bars called stirrups placed perpendicular to the cracks. This experimental study aims at exploring ways of making the use of 3mm wires as stirrups in concrete beams, with various spacing so as to make the stirrup efficient and cost-effective for cheaper construction. The 3mm wire is made continually spiral, to avoid the unwinding of stirrups. The plain concrete beam with spiral stirrups are compared with that of the standard beam with 2-legged stirrup and the analysis is done using the Abaqus software to know the variations in load versus displacement curve, maximum principal stress, minimum principal stress between the normal stirrup and the continuous spiral stirrup.**

Keywords: **Normal stirrups, spiral stirrups, 3mm wire, FEA**

1. INTRODUCTION

A beam is a structural element that is capable of withstanding load primarily by resisting bending. The bending force induced into the material of the beam as a result of the external loads, own weight, span and external reactions to these loads is called a bending moment. Beams generally carry vertical gravitational forces but can also be used to carry horizontal loads (i.e., loads due to an earthquake or wind).

Successful performance of reinforced and prestressed concrete members requires an effective interaction between concrete and reinforcing steel. Not only is an adequate amount of reinforcement needed, but it must also be properly detailed to insure satisfactory member behaviour under all loading conditions.

Beams sustain two basic types of failures, namely:

(a) Flexural (or bending) Failure

(b) Shear Failure

Flexural (or bending) Failure:

As the beam sags under increased loading, it can fail in two possible ways. If relatively more steel is present on the tension face, concrete crushes in compression; this is a brittle failure and is therefore undesirable. If relatively less steel is present on the tension face, the steel yields first and redistribution occurs in the beam until eventually the concrete crushes in compression; this is a ductile failure and hence is desirable. Thus, more steel on tension face is not necessarily desirable! The ductile failure is characterized with many vertical cracks starting from the stretched beam face, and going towards its mid-depth. 2

ISSN 2394-7357

International Journal of Novel Research in Civil Structural and Earth Sciences
Vol. 6, Issue 3, pp: (14-28), Month: September - December 2019, Available at: www.noveltyjournals.com

(b) Shear Failure:

A beam may also fail due to shearing action. A shear crack is inclined at 45° to the horizontal; it develops at mid-depth near the support and grows towards the top and bottom faces. Closed loop stirrups are provided to avoid such shearing action. Shear damage occurs when the area of these stirrups is insufficient. Shear failure is brittle, and therefore, shear failure must be avoided in the design of RC beams.

When a Reinforced Concrete beam subjected to a combination of moment and shear force with either little or no transverse reinforcement can fail prematurely in shear before reaching its full flexural strength. This type of shear failure is sudden in nature and usually catastrophic because it doesn’t give ample warning to inhabitants. According to IS456:2000 to prevent shear

Failure beams should be reinforced with stirrups.

Usually stirrups provided in beams are in the form of two-legged stirrups. In two-legged stirrups cause unwinding of stirrups due to loading acting on beam. This will affect the shear strength of the beam. So to avoid this type of failure in beams, stirrups are provided continuously or spirally. Nowadays spiral or helical reinforcement is used in columns only. So the helical reinforcement in beams is used in this paper. The actual method and proposed method for stirrups used in beams is shown in below.

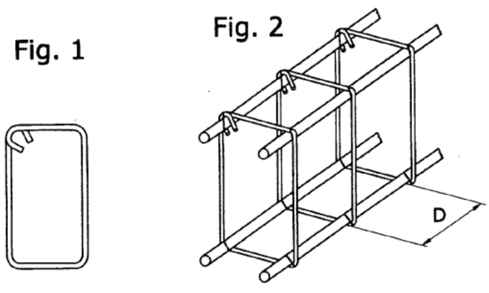

Figure 1.1: Actual method of winding stirrups

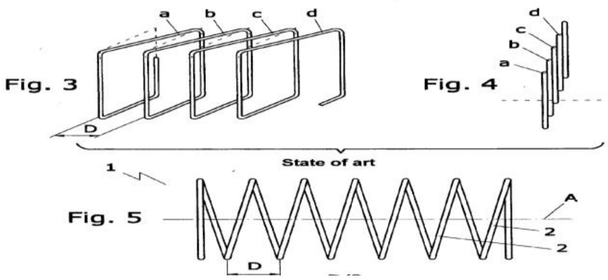

Figure1.2: Proposed method of winding stirrups

Successful performance of reinforced and prestressed concrete members requires an effective interaction between concrete and reinforcing steel. Not only is an adequate amount of reinforcement needed, but it must also be properly detailed to insure satisfactory member behaviour under all loading conditions. Those stirrups are provided in separate units in beam. So, we tried to provide continuous stirrups like columns.

On a brief note from the papers, the performance of stirrups was improved by avoiding the use of single-legged stirrups. The shear crack width of beam proportionally increases with the strain of shear reinforcement and the spacing between the shear cracks. Then, the beam size did not affect the compatibility of the transverse reinforcement to provide shear resistance and also the beam size did not affect the post cracking behaviour or the shear strength provided by the stirrups. . When the four-leg stirrups were used, the efficiency of the beam to resist the shear force was increased with a constant amount of stirrups than two-leg stirrup configuration even when a smaller area of stirrups was used. And also the bar diameter had no significant influence on the performance of the stirrups.

International Journal of Novel Research in Civil Structural and Earth Sciences
Vol. 6, Issue 3, pp: (14-28), Month: September - December 2019, Available at: www.noveltyjournals.com

MIX DESIGN OF CONCRETE

The beams are casted using M25 concrete. Two types of concrete mixes are used here. One mix contains 20mm coarse aggregate and another mix contains 8mm chips as a coarse aggregate. Because in continuous stirrups, pouring of 20mm coarse aggregate concrete in 500x100x100 size beam is difficult. So we used 8mm coarse aggregate mix.

Table1.1: Mix proportions of concrete

WATER	CEMENT	COARSE AGGREGATE	FINE AGGREGATE
191.6	383	546	1187
0.5	1	1.42	3.09

As per two point load method and IS456;2000code, 500X 100X 100 size beam is designed. The spacing of normal stirrups as per design is 47mm and the spacing of normal stirrups is 14mm.

2. METHODOLOGY ON WINDING OF STIRRUPS

The stirrups are generally performing the dual role in beam. That is increase the shear strength of the beam and reduces the shear crack width. These stirrups provided in different ways like single-legged stirrups, double-legged stirrups and four-legged stirrups etc. and these are all perform a separate unit in beam. In circular columns we see the continuous spiral stirrups to increase the shear strength But in beams e didn't use any continuous spiral stirrups. So, we tried continuous spiral stirrups in beams.

The transverse reinforcement in beam using 3mm wire is arranged in different methods to check the effectiveness of spiral reinforcement in beam. In first method, 3 beams are prepared by using 3mm wire, which is winding around the main reinforcement as continuous direction or spirally with equal spacing of 56mm. The prepared models are shown below.

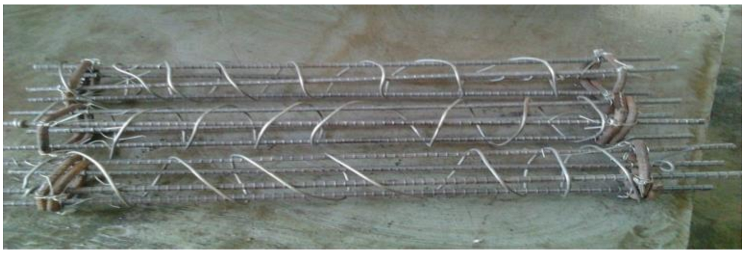

Figure 2.1: Circular spiral stirrup beam

In second method, wooden mould is used for the winding process. Because, the continuous spirals used in first method are not accurate shape. But, the spiral ring which is getting by wooden mould is not fitted over the main reinforcement. So, this method is not suitable for spiral reinforcement.

In third method, the 3mm wire is winding over the reinforcement directly. As per design 24 spirals and 3 stirrups are provided in this model. The second model is prepared with 32 spirals and 3 stirrups. The process of winding and the whole model figures are shown below.

Figure 2.2: Reinforcement bar before winding

International Journal of Novel Research in Civil Structural and Earth Sciences
Vol. 6, Issue 3, pp: (14-28), Month: September - December 2019, Available at: www.noveltyjournals.com

Figure 2.3: Process of winding

Figure 2.4: Continuous spiral reinforcement

In last method, 10 stirrups are provided with equal spacing of 47mm in beam as per design. It is used to compare the results with normal stirrups to spiral stirrups.

Figure 2.5: Beam with normal 6mm stirrups

Table 2.1: Specimen details

PARTICULARS	LENGTH	BREADTH	DEPTH	STIRRUPS
Beam (NS)	500	100	100	6mm @ 47mm
Beam(GI)	500	100	100	3mm @ 14mm
Beam(GI)	500	100	100	3mm @18mm
Beam(GI)	500	100	100	3mm @ 56mm

3. CASTING AND TESTING OF BEAMS

The beam moulds of size 500 x 100 x 100 are used for casting. The materials required for casting is weighed as per mix design and mix in correct proportions. Here two types of mixes used for casting. The one mix contains 20mm angular coarse aggregate and another mix contains 8mm chips as a coarse aggregate. The 20mm coarse aggregate mix is used for the beams with 8 spirals and the 8mm chips mix is used for the beams with 24, 32 spirals and normal stirrups.

Figure 3.1: casting of beam with normal stirrups and 3mm wire spiral stirrups

The testing of beams was conducted in the same testing machine but by means of using flexural setup. The 2point loading has been done by considering the specimen into 3 divisions. The loading was given at the rate of 0.5kN/sec and the readings at the failure were taken. The failure pattern is shown below and the obtained data is tabulated.

Figure 3.2: Formation of cracks in beam

4. FINITE ELEMENT ANALYSIS

Finite element analysis is a computer based numerical technique for calculating the strength and behaviour of engineering structures. It is used in new product design, and existing product refinement. The essence of a finite element solution of an engineering problem is that a set of governing algebraic equations is established which are then solved with the help of a digital computer. Analytical investigations allow creating models which help in parametric studies for design and later provide the base for future studies on similar systems without going into expensive and time-consuming experimental studies. There are different analytical models available, among that models, finite element analysis-FEA, has proved it to be a solid tool for various kinds of analyses. The purpose of a finite element analysis is to re-create mathematically the behaviour of an actual engineering system. By analytical investigation, the analytical model of the physical experiments carried out in the laboratory is created. It enables engineers to build CAD models of structures and analyse complex structural systems. This model comprises of all the nodes, elements, material properties, real constants, boundary conditions and other features that are used to represent the physical system.

A wide range of commercially available software like DIANA, ADINA, ATENA, ANSYS and ABAQUS etc are used for this purposes. In that ABAQUS has been used 1.5m beam model.

MODEL SIMULATION:

A complete Abaqus analysis usually consists of three distinct stages: pre-processing, simulation, and post processing. These three stages are linked together by files as shown below:

International Journal of Novel Research in Civil Structural and Earth Sciences
Vol. 6, Issue 3, pp: (14-28), Month: September - December 2019, Available at: www.noveltyjournals.com

1) Pre-processing (Abaqus/CAE): In this stage we must define the model of the physical problem and create an Abaqus input file. The model is usually created graphically using Abaqus/CAE or another pre-processor, although the Abaqus input file for a simple analysis can be created directly using a text editor.

2) Simulation (Abaqus/Standard or Abaqus/Explicit): The simulation, which normally is run as a background process, is the stage in which Abaqus/Standard or Abaqus/Explicit solves the numerical problem defined in the model. Examples of output from a stress analysis include displacements and stresses that are stored in binary files ready for post-processing. Depending on the complexity of the problem being analysed and the power of the computer being used, it may take anywhere from seconds to days to complete an analysis run.

3) Post-processing (Abaqus/CAE): We can evaluate the results once the simulation has been completed and the displacements, stresses, or other fundamental variables have been calculated. The evaluation is generally done interactively using the Visualization model of Abaqus/CAE or another postprocessor. The Visualization model, which reads the neutral binary output database file, has a variety of options for displaying the results, including colour contour plots, animations, deformed shape plots, and *X–Y* plots.

FINITE ELEMENT MODELLING

Abaqus/CAE is divided into functional units called models. Each model contains only those tools that are relevant to a specific portion of the modelling task. We can select a model from the Model list in the context bar. Alternatively, we can select a model by switching to the context of a selected object in the Model Tree.

Part:

Parts created in Abaqus/CAE are constructed from an ordered list of features and the parameters that define the geometry of each feature. Using the tools in the Part model, we create and edit all the features necessary to describe each of the parts in our model. Abaqus/CAE stores each feature and use this information to define the entire part, to regenerate the part if we modify it. Here two different models are created. One is beam with normal stirrups and other is beam with 3mm spiral stirrups. Modelling of the composite beam consists of two parts including Concrete and Steel. The concrete part is created in 3D modelling space with deformable type. Basic feature of this part is extruded type solids.GI-wire spiral Stirrups (3mm) of the structure are modelled with solidworks software and imported in parts.

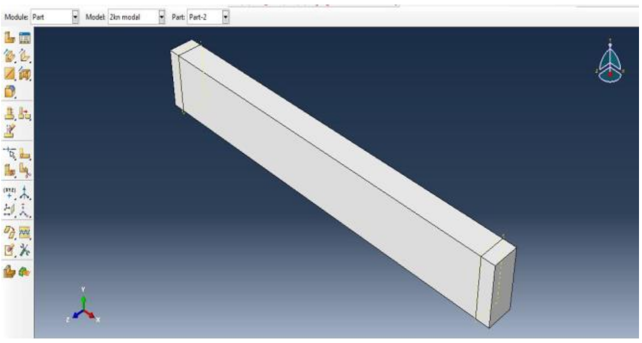

Figure 4.1: Part –Beam

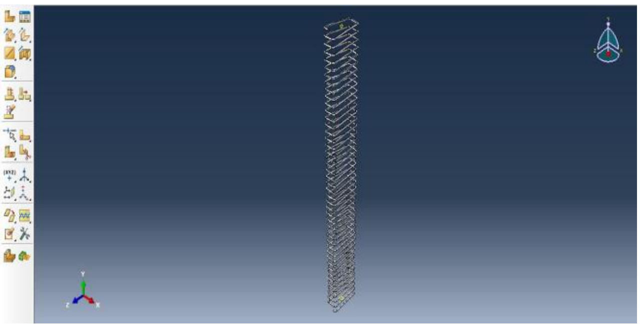

Figure 4.2: Part-3mm spiral stirrups

Novelty Journals

ISSN 2394-7357

International Journal of Novel Research in Civil Structural and Earth Sciences

Vol. 6, Issue 3, pp: (14-28), Month: September - December 2019, Available at: www.noveltyjournals.com

Figure 4.3: Part-6mm normal stirrups

Material model:

The material model is used to assign the properties of materials involve in the model. In this two models concrete and steel are the materials to assign the property. The general properties of concrete and steel are given below.

Table 4.1: Material properties

MATERIAL	ELASTIC MODULUS N/mm^2	DENSITY N/mm^3	POISSON'S RATIO
CONCRETE	25000	24000	0.18
STEEL	200000	77000	0.3

Concrete damaged plasticity

The concrete damaged plasticity model in Abaqus provides a general capability for modelling concrete and other quasi-brittle materials in all types of structures (beams, trusses, shells, and solids). It can be used for plain concrete, even though it is intended primarily for the analysis of reinforced concrete structures and it is also used with rebar to model concrete reinforcement. The concrete damaged plasticity of this model is shown below.

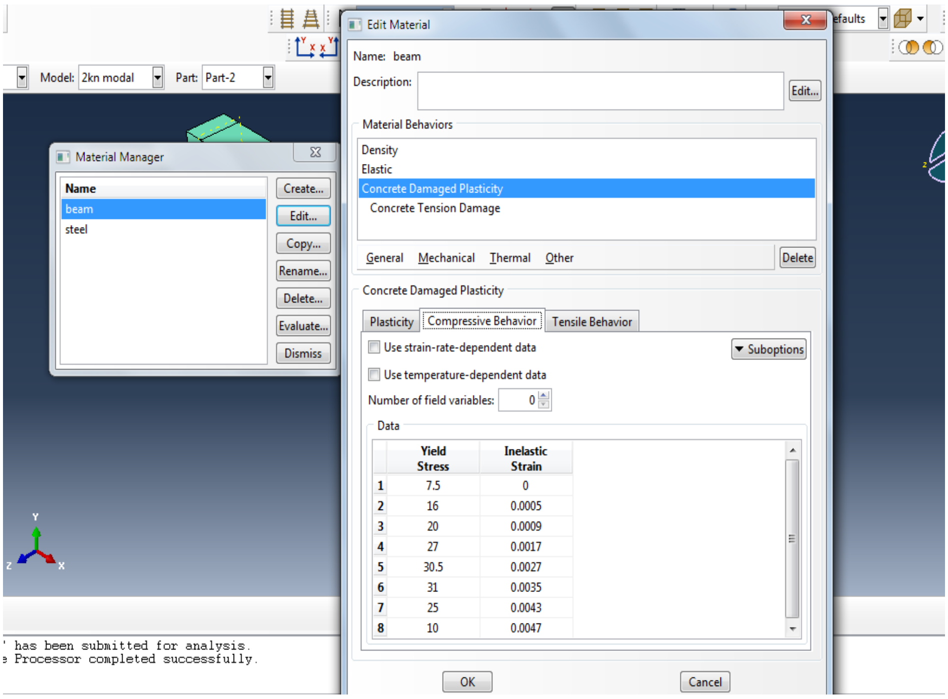

Figure 4.4: Concrete Damaged Plasticity

Using material and section creation tabs the material properties are assigned to its section. Homogeneous solid sections are created. With the help of assign section tab assigned this properties to the composite structure.

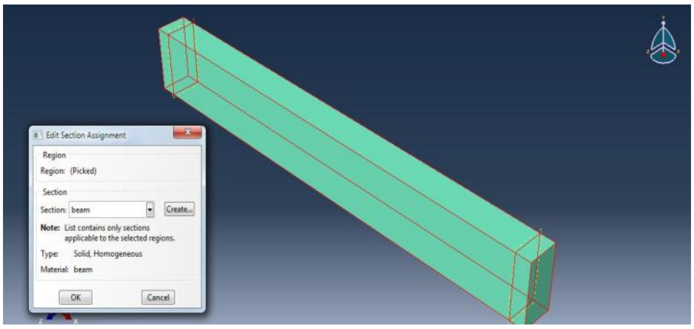

Figure 4.5: Assigning Section to the Material

Geometrical model

The creation of continuous stirrup in ABAQUS is very difficult. So the continuous stirrup in rectangle spiral is created in SOLIDWORKS software. The creation of rectangle spiral is done in step by step.

The first step is creating rectangle shape with the dimensions of 160x60. Because the effective depth of beam is 160mm and the width is 60mm.Then extrude the rectangle shape with the length of 1450mm.

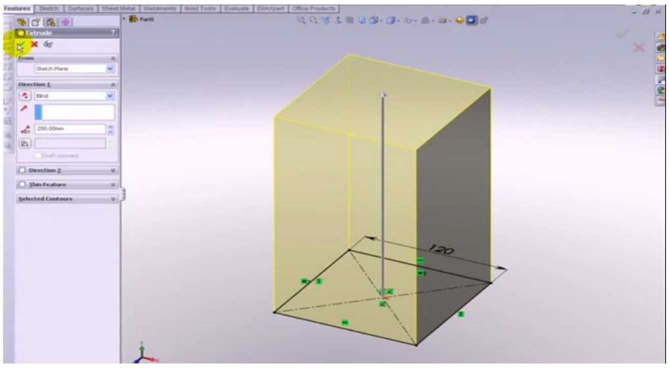

Figure 4.6: Extruded Rectangle in Solidworks

Then the bottom of the rectangle is selected and creates the circle in center. The circle is act as the origin of stirrup. Next we go to features option, in that select curve. Then edit the parameters of the spring like pitch distance, number of spirals and rotation angle. The spring will generate in the rectangle automatically. The next step is going to the front view of the rectangle box and draws a line horizontally, which is connected to the spring by coincident option. Then using sweep surface option, the spring will be swept. In that the select the line as a profile and select the spring as a path. Then hide the surface sweep and add the fillets at the corners of the rectangle box and unhide the surface sweep. The next step is creating intersection curve. In this step select the surfaces of the rectangle box and the swept surface. The intersection curve will be generated. Then delete the rectangle shape and others. The rectangle spiral was created.

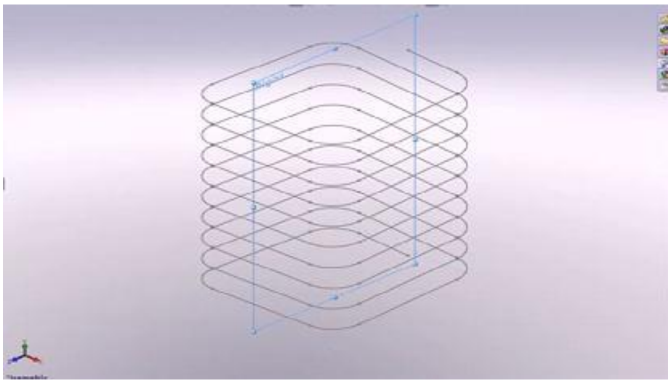

Figure 4.7: Rectangular Spiral Ring in Solidworks

The next step is to create the circle at the end of the spring as 3mm and using sweep option the spring is changed to continuous spirals.

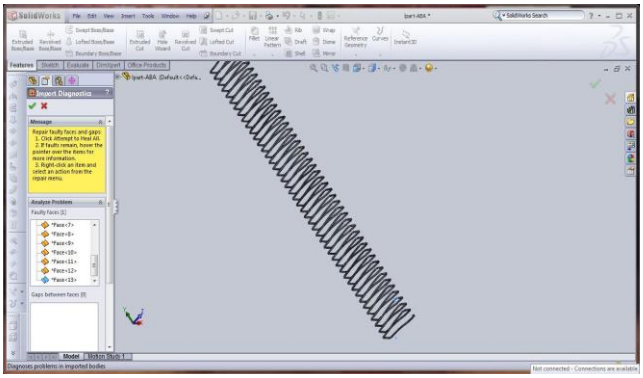

Figure 4.8: Swept Rectangular Spiral Ring

Then the solidworks file saved in IGES format. The creation of rectangle spring is done. Then open the ABAQUS file and import the solidworks file as a part. Now, the spring will act as a part of the model.

Mesh:

The Mesh module allows us to generate meshes on parts and assemblies created within Abaqus/CAE. The Mesh module provides the tools for prescribing mesh density at local and global levels and Model colouring that indicates the meshing technique assigned to each region in the model.

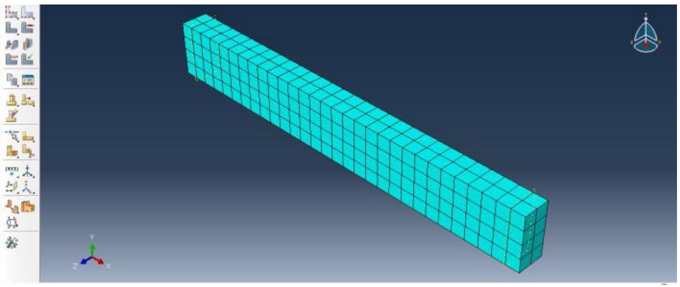

Figure 4.9: Mesh- Beam

Assembly:

Assembly of the part is done in this step. When we create a part, it exists in its own coordinate system, independent of other parts in the model. In contrast, we use the Assembly model to create instances of our parts and to position the instances relative to each other in a global coordinate system, thus creating the assembly. We can position part instances by sequentially applying position constraints that align selected faces, edges, or vertices or by applying simple translations and rotations. Loads, boundary conditions, predefined fields, and meshes are all applied to the assembly. An instance maintains its association with the original part. In first model concrete block and normal stirrups are assembled. In second model, concrete block and 3mm spiral stirrups are assembled. They are shown below

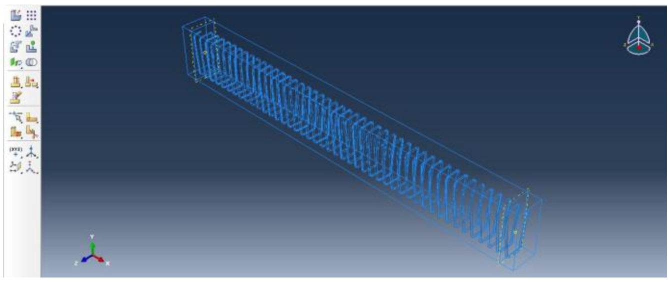

Figure 4.10: Assembly-beam with continuous stirrups

Step:

Within a model we define a sequence of one or more analysis steps. The step sequence provides a convenient way to capture changes in the loading and boundary conditions of the model, changes in the way parts of the model interact with each other, the removal or addition of parts, and any other changes that may occur in the model during the course of the analysis. In addition, steps allow us to change the analysis procedure, the data output, and various controls. We can also use steps to define linear perturbation analyses about nonlinear base states. We can use the replace function to change the analysis procedure of an existing step. In this process we can use two different types of analysis steps. They are 1.Static, General method 2.Static, Riks method

Load module:

Load and boundary condition are given in this step. Load is given as concentrated forces like a two point load. Support is given at the edges. Support at one end is hinged and the support at another end is roller support.

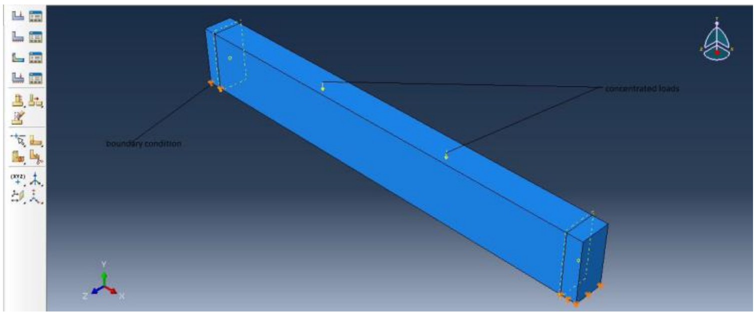

Figure 4.11: Load model

Job:

The Job module allows us to create a job, to submit it to Abaqus/Standard or Abaqus/Explicit for analysis, and to monitor its progress.it allows us to submit more files and run that at a time. In addition, we have the option of creating only the analysis input file for model. This option allows us to view and edit the input file before submitting it for analysis. The results of the model show stress, strain and deflection of beam at loading.

5. RESULTS AND DISCUISSIONS

Tests were conducted to evaluate the shear carrying capacity of RC with normal 6mm stirrups and RC with 3mm spiral stirrups and its failure behaviour. The behaviour of specimens in each test and comparison of the experimental results

EXPERIMENTAL RESULTS:

Figure 5.1: Failure pattern of specimen A

Novelty Journals

ISSN 2394-7357

International Journal of Novel Research in Civil Structural and Earth Sciences
Vol. 6, Issue 3, pp: (14-28), Month: September - December 2019, Available at: www.noveltyjournals.com

Figure 5.2: Failure pattern of specimen B

Figure 5.3: Failure pattern of Specimen C

Figure 5.4: Failure pattern of Specimen D

Table 5.1: Experimental results

PARTICULARS	1ST CRACKING LOAD	ULTIMATE CRACKING LOAD	FAILURE PATTERN
Beam with normal 6mm stirrups (A)	36kN	52.65kN	Shear failure
Beam with 8 spirals of 3mm GI wire(B)	30.20kN	51.23kN	Shear failure
Beam with 24 spirals of 3mm GI wire (C)	41.23kN	62.05kN	Shear & flexure failure
Beam with 32 spirals of 3mm GI wire (D)	35.26kN	59.12kN	Shear & flexure failure

DISCUISSION OF EXPERIMENTAL RESULTS

The resulting failure loads of specimens with normal stirrups and continuous stirrups are summarized in Table 8. The failure loads of the specimens are 52kN (normal stirrups), and 51kN (GI-8spirals), 62kN (GI-24 spirals), 59kN (GI-32 spirals). Based on the testing results, the beam reinforced with continuous stirrups are more effective than the beam reinforced with normal stirrups.

RESULTS FROM ANALYSIS:

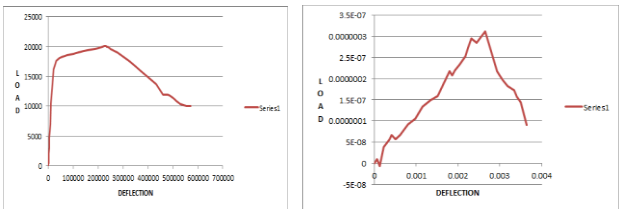

Figure 5.5: Comparison of load vs. deflection curve for normal stirrups reinforced beam to spiral stirrup reinforced beam

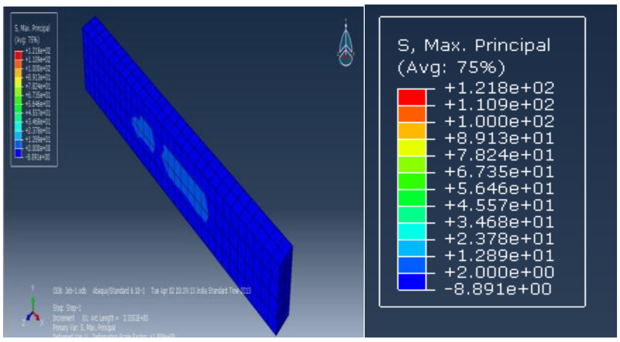

Figure 5.6: Maximum principal stress for normal stirrup beam

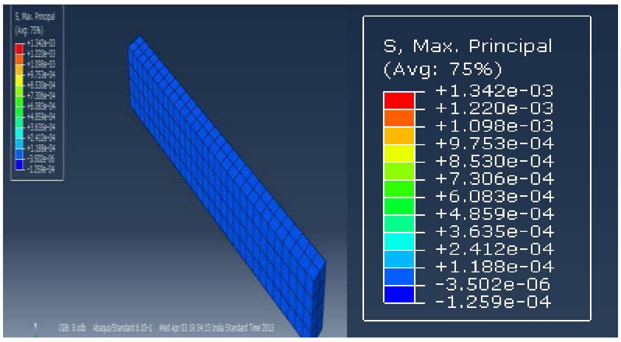

Figure 5.7: Maximum principal stress for spiral stirrup beam

ISSN 2394-7357
International Journal of Novel Research in Civil Structural and Earth Sciences
Vol. 6, Issue 3, pp: (14-28), Month: September - December 2019, Available at: www.noveltyjournals.com

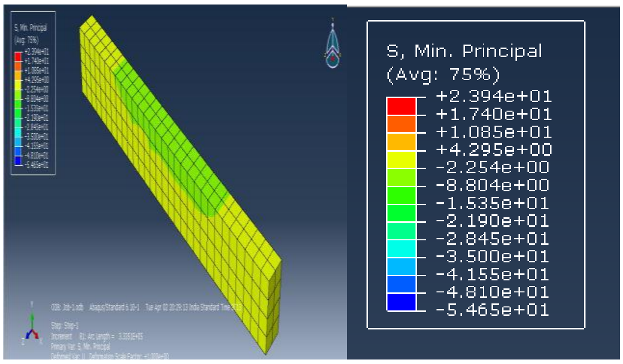

Figure 5.8: Minimum principal stress for normal stirrups beam

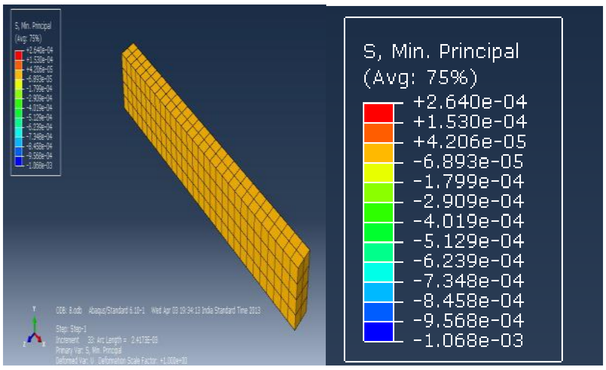

Figure 5.9: Minimum principal stress for spiral stirrup beam

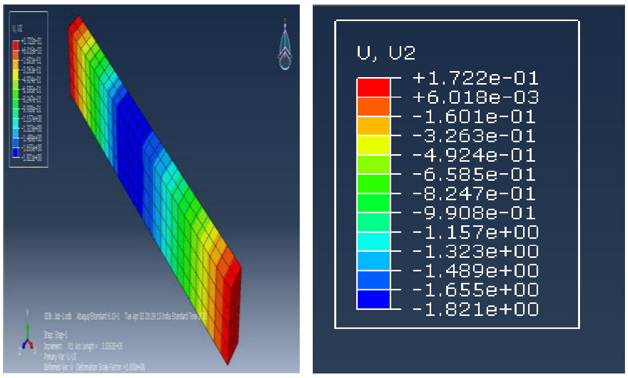

Figure 5.10: Deflection of beam with normal stirrups beam

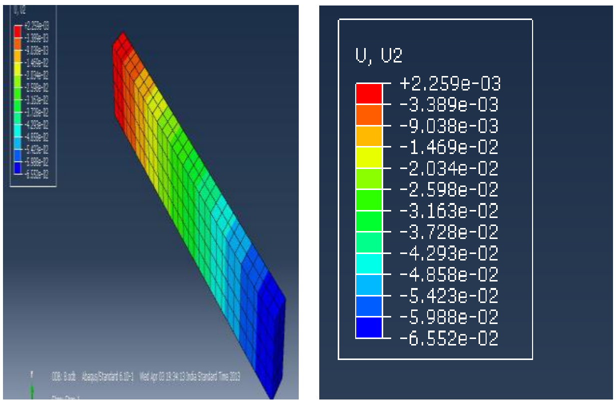

Figure 5.11: Deflection of beam with spiral stirrup beam

From the above results from ABAQUS shows the spiral stirrups reinforced beams are more effective than the normal reinforced beam.

6. CONCLUSIONS

Based on the research made with the continuous stirrup in beam, the following conclusions are made,

- The shear strength of the beam which is reinforced with continuous spiral stirrups increases by nearly 10kN.
- The beam with rectangular spiral stirrup is more effective than the beam reinforced with circular spiral stirrup.
- In finite element analysis, the deflection of beam is very less in spiral stirrups beam when compared to normal stirrups beam.
- The minimum principal stress and maximum principal stress are very less in spiral stirrups beam than the normal stirrups beam.

REFERENCES

[1] Altoubat.S, Yazdanbakhsh.A, and K.Alexander Rieder (2009),"Shear Behaviour of Macro-Synthetic Fiber-Reinforced Concrete Beams without Stirrups", American Concrete Institute Materials Journal, Vol- 106, No. 4, July-August 2009, pp 381-389.

[2] Anderson.N.S, Ramirez.J.A. (1989),"Detailing of Stirrup Reinforcement" ACI Structural Journal, Sep-Oct 1989, Vol. 86, No.5, pp 507-515.

[3] Anil Kumar.R, R. RudraPrasad," Comparison study between steel Fiber Reinforced Concrete Beams without Stirrups and Reinforced Concrete beams with stirrups", Lecturer of Civil Engineering, Dayananda Sagar College of Engineering.

[4] Be I," he Shear Strength of Concrete Containing Fiber-Reinforced Plastic (FRP) Reinforcement" Lecturer, Dept. o Architecture and Civil Engineering, University of Bath, United Kingdom.

[5] Eberhard.M.O, Woo-Suk Kim, Jubum Kim and Yoon-Keun Kwak (2002),"Shear Strength of Steel -Fiber Reinforced Concrete Beams Without Stirrups", ACI structural journal , Vo -99, No.4, July-August 2002, pp 530-538.

[6] El-Sayed.A.K, El-Salakawy and Benmokrane.B (2007)," Mechanical and Structural Characterization of New Carbon FRP Stirrups for Concrete Members", Journal of Composites for Construction, Vol. 11, No.4, August 1, 2007. ISSN 1090-0268/2007, pp 352–362.

[7] Emi e Shehata," Use of FRP as Shear Reinforcement for Concrete Structures", ISIS Canada Network of Centres of Excellence, University of Manitoba, Winnipeg, Manitoba, Canada.

[8] Frosch.R.J (2000),"Behaviour of Large-Scale Reinforced Concrete Beams with Minimum Shear Reinforcement" American Concrete institute Journal, Vol-97,No.6,Nov-Dec 2000. Title no.97-S82, pp 814-820.

[9] Hsiumg.W and G.C. Frantz (1985)," Transverse Stirrup Spacing in R/C Beams", ASCE, Journal Of Structural Engineering, Vol-1ll, No. 2, February, 1985, ISSN 0733-9445. Paper No. 19515, pp 353-362.

[10] J.C. Varney, M.D.Brown, O.Bayrak, and R.W. Poston (2011),"Effect of Stirrup Anchorage on Shear Strength of Reinforced Concrete Beams"" ACI Structural Journal, July-August 2011, Title no. 108-S45, pp 469-478.

[11] Kobraei.M M, Jumaat.M.Z M Z and Shafigh.P (20),"An experimental study on shear reinforcement in RC beams using CFRP-bars" Scientific Research and Essays Vol. 6(16), 19 August 2011, pp. 3447-3460.

[12] Maruthachalam.D, Karthik.M.P (2013), "Shear Performance of Fiber Reinforced Concrete Beams", Indian Journal of Engineering, Volume 2, Number 3, January 2013.

[13] Mohamed Zakaria, Tamon Ueda, Zhimin Wu, Liang Meng (2009),"Experiments In estimation On Shear Crack in Behaviour in Reinforced Concrete Beams With Shear Reinforcement", Journal of Advanced Concrete Technology, Feb-2009, Vol-7,No.1, pp79-96.

International Journal of Novel Research in Civil Structural and Earth Sciences
Vol. 6, Issue 3, pp: (14-28), Month: September - December 2019, Available at: www.noveltyjournals.com

[14] Qiang Yu, Bažant Z P and Hon M (20),” Can Stirrups Suppress Size Effect on Shear Strength of RC Beams?” ASCE Journal of Structural Engineering, Vol. 137, No. 5, May 1, 2011. pp 607– 617.

[15] Ryan Morphy, ”Performance o FRP as Shear Reinforcement or Concrete Structures”, President, ISIS Canada; Professor, Dept. of Civil Engineering, University of Manitoba, Canada.

[16] Shuraim A B,” Transverse Stirrup Configurations in RC Wide Shallow Engineering, Vol- 138, No. 3, March 1, 2012, ISSN 0733-9445/2012,pp 416–424.

[17] Tim Stratford, Chris Burgoyne (2003), "Shear Analysis of Concrete with Brittle Reinforcement" Journal of Composites for Construction, Vol- 7, No. 4, November 1, 2003. ASCE, ISSN 1090-0268/2003, pp 323–330.

[18] Tompos E Frosch R J (2002),”Influence O Beam Size, Lon gitudinal Reinforcement and Stirrup Effectiveness in Concrete Shear Strength” ACI Structural Journal, Vol-99, No.5, Sep-Oct 2002, pp 559-567.