International Journal of Applied Environmental Sciences
ISSN 0973-6077 Volume 12, Number 1 (2017), pp. 201-210

http://www.ripublication.com

Numerical Analysis of High Strength Concrete Beams using ABAQUS

Sethuraman V.S[1], Suguna K[2] and Raghunath P.N[2]

[1] Research Scholar, Department of Civil & Structural Engineering, Annamalai University, Annamalainagar, Tamilnadu-608002

[2] Professor, Department of Civil & Structural Engineering, Annamalai University, Annamalainagar, Tamilnadu-608002

**Corresponding author's*

Abstract

The objective of this paper is to model and analyses an M60 concrete Beam using Abaqus for static load and verify the same using experiment. In recent years, Concrete having a compressive strength of 60 MPa and above is being used for high-rise buildings and long span bridges. The advent of various mineral and chemical admixtures has facilitated the production of high strength to high performance concrete. HSC helps in avoiding the use of unacceptable oversized columns on the lower floors, allowing large column spacing and usable floor space or increasing the number of possible stories without detracting from the lower floor. HSC reduces the dead load of bridge girders and piers enabling larger underpass clearance widths. ABAQUS is a suite of powerful engineering simulation programs, based on the finite element method (FEM) that can solve problems ranging from relatively simple linear analyses to the most challenging nonlinear simulations. In this paper an M 60 beam has been modeled with different reinforcement configurations and results has been compared.

Keywords: Abaqus, FEM, High strength concrete beam, Compressive strength

INTRODUCTION:

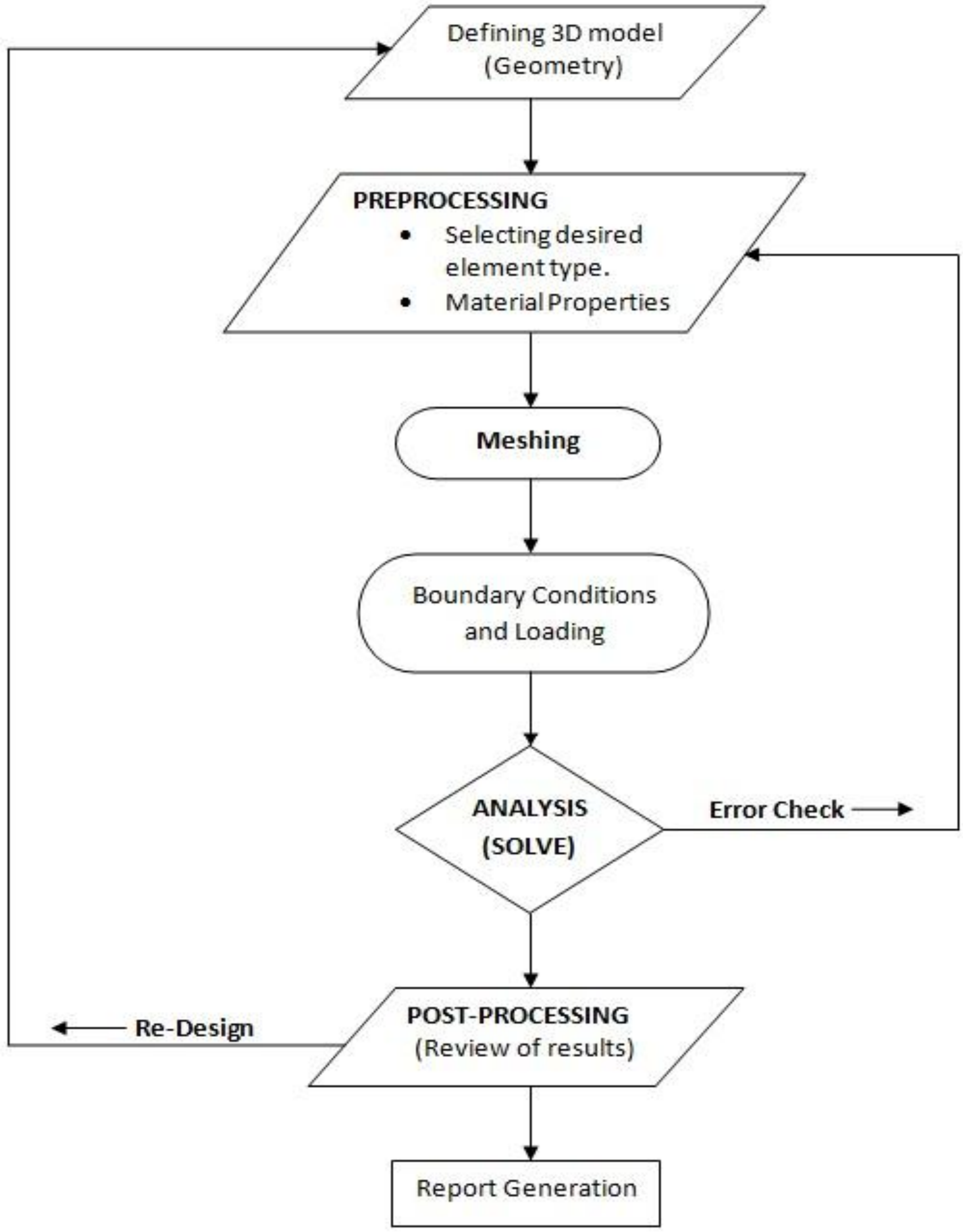
The advent of newer concrete making technologies has given impetus for making concrete of higher strength. As per our Indian standard IS 456: 2000 concretes are grouped as ordinary concrete, standard concrete and high strength concrete as given in Table 1. The code did not describe about UHSC, but the American Concrete Institute (ACI) categories the concrete as Normal Strength Concrete (NSC), High Strength Concrete (HSC) and Ultra High Strength Concrete.

Table 1. Group of Concrete as per IS 456:2000

Sl. No	Name of Group of Concrete	Grade Designation
1	Ordinary Concrete	M10 to M20
2	Standard Concrete	M25 to M55
3	High Strength Concrete	M60 to M80
Note: M refers to mix and the number to specified compressive strength of 150 mm size cube at 28 days expressed in N/ mm2		

Closed-form solutions for the analysis offer in forced structural members are normally based online are elastic models. These are not capable of dealing with problems where gross material and geometric non-linearity's exist. It is desirable to predict effects such as strain and stress variations within an RC beam with steel reinforcement while undergoing non-linear changes. One of the main approximations associated with non-linear behavior of concrete is the modeling of concrete cracking. Under the application of load, concrete cracks in the tensions one and as a result the stress path becomes discontinuous and the load transfer changes at the cracked section. Adoption of appropriate material criteria and concrete elements that would model discrete cracking of concrete is an essential requirement. Infinite element analysis, various procedures have been adopted for predicting cracking in concrete. Smeared and discrete crack formulations are quite common. In smeared crack approach, cracks are simulated as local discontinuities which are smeared within the finite element; in discrete crack approach, cracks are introduced in the finite element model using interface elements between the concrete ones. The problem with smeared crack approach is that it tends to spread crack formation over the entire structure, which makes it difficult to predict localized failures. In spite of this short coming, the method is widely used for precisely predicting the load deflection and load strain behavior of concrete. The problem with discrete crack approach is that the position and direction of crack growth is predefined. However the method is being used for predicting the non-linear effects in concrete. The most commonly used finite element codes such as ABAQUS, ANSYS, LUSAS, DIANA and ADINA have more versatile material models capable of modeling concrete.

STEPS IN SIMULATION:



The ABAQUSfinite element program was used in this study to simulate the behavior of the experimental beams. ABAQUS/Standard was selected for this simulation, since its interface is very easy to use and supports parametric modelling. Geometry of theconcrete beam was created using ABAQUS/CAE and the element type was applied to geometry by command prompt. Reinforcement in a concrete beam was created as 1D beam model with cross section specified in ABAQUS/CAE.

ELEMENT TYPES

Element type used for this study is listed in the table below. C3D8 element type gives more stable results.

Table 1.3: ElementTypesforWorkingModel

Material Type	Element
Concrete	C3D8
Steel Reinforcement	BEAM

MATERIAL PROPERTIES

ABAQUS has vas set of material library in the engineering data section. Either we can select a material from the library or we can manually enter the properties of material in ABAQUS/CAE. The C3D8element requires linear isotropic and multi-linear isotropic material properties to properly model concrete. EX is the modulus of elasticity of the concrete (E*c*), and PRXY is the Poisson's ratio (μ). The material model in Aaqus/CAE requires that different constants be defined. Those 9 constants are shear transfer coefficients for an open crack, shear transfer coefficients for a closed crack, uniaxial tensile cracking stress, uniaxial crushing stress (positive), Biaxial crushing stress (positive), ambient hydrostatic stress state for use with constants 7 and 8, biaxial crushing stress (positive) under the ambient hydrostatic stress state, (constant6), Uniaxial crushing stress (positive) under the ambient hydrostatic stress state (constant6), stiffness multiplier for cracked tensile condition. Fig 1.shows modeling of HSC beams

Table 1.4: MaterialModelsfor C3D8

Linear Isotropic			
EX	34,948 Mpa	34,948 Mpa	34,948 Mpa
PRXY	0.20	0.20	0.20

PREPROCESSING IN ABAQUS

The concrete and reinforcements has been modeled in Abaqus

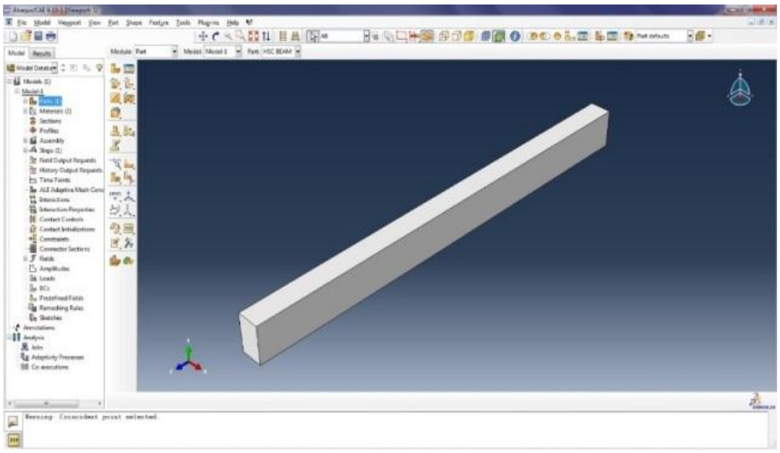


Figure 1.a: Modeling of Concrete Beam

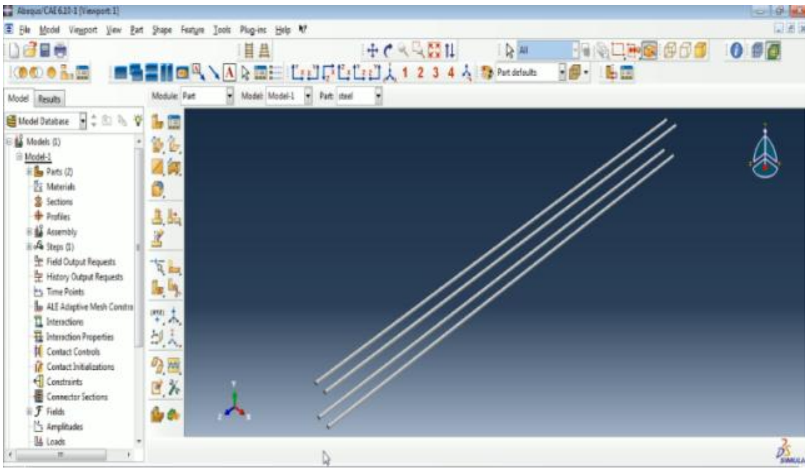


Figure 1.b: Modeling of Reinforcement

MESHING

Meshing plays a vital role in the FEA since the properties and governing relationships are assumed over the discretized elements and expressed mathematically on the specified points called nodes. Hence increasing the number of elements in a Finite element model will increase accuracy but at the same point it will take more time to solve the equations. The below figure show the meshed models with solid C3D8 element and Beam element. Figure 1 shows modelingof High Strength Concrete Beam.

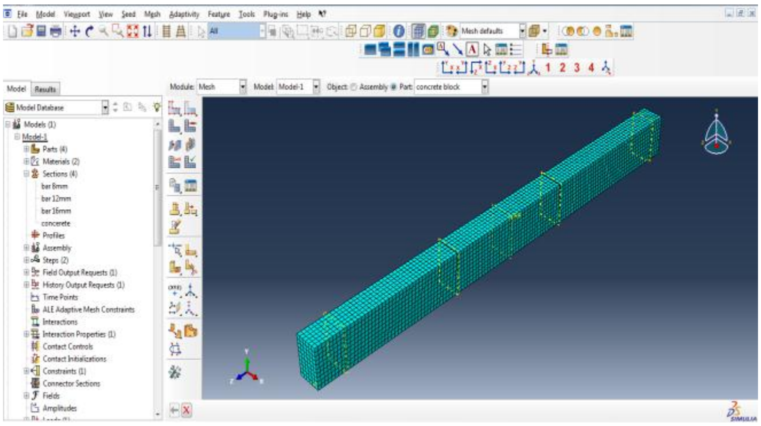


Figure 1.c: Meshed Concrete Beam

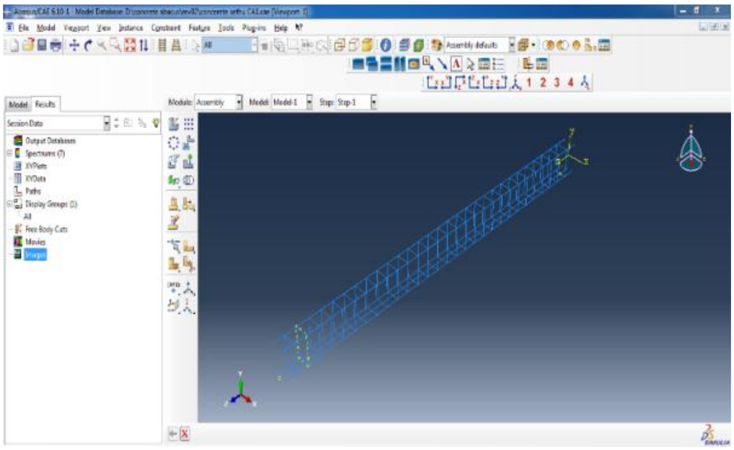


Figure 1.d: Meshed Reinforcement with Cross Section

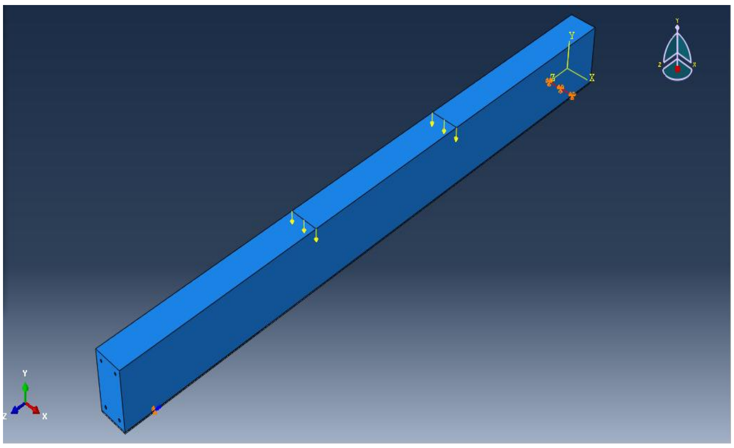


Figure 1.e: Defining Load

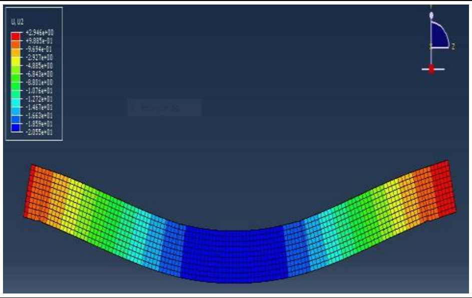
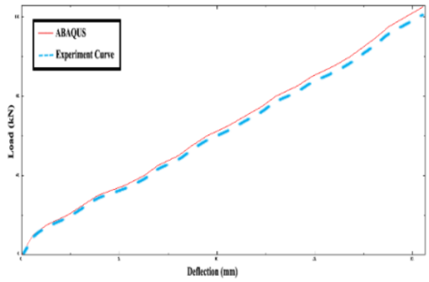
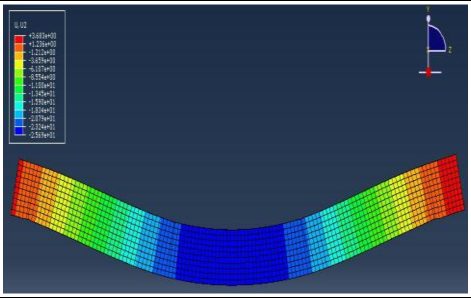
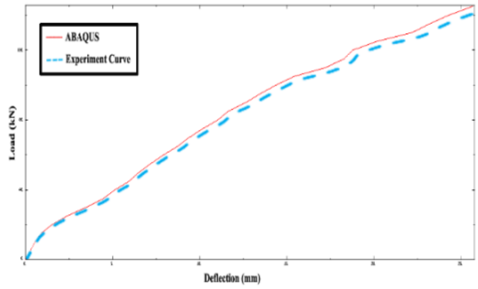
Figure 1. Modeling of concrete beams

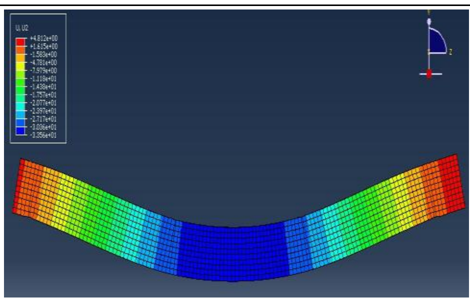
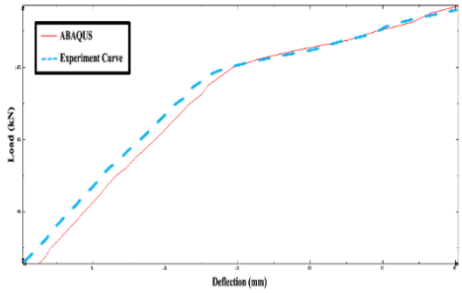
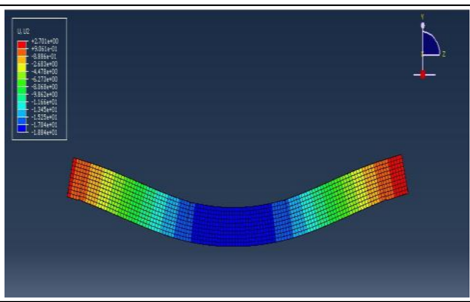
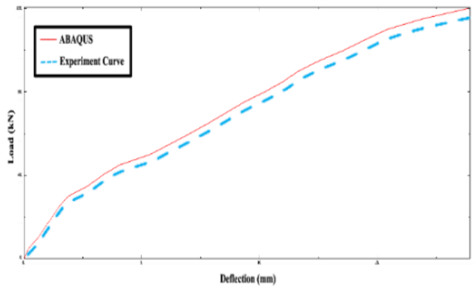
RESULTS AND DISCUSSION OF FINITE ELEMENT ANALYSIS

The six no’s of beam specimens tested under static loading and six no’s of beam specimens tested under cyclic loading were analyzed using the ABAQUS Standard. The results pertaining to the objectives of the study arepresented and discussed in this section. The finite element analysis results of the concrete beams specimens at different load levels are presented in Table 2.The six no’s of beam specimens were tested under static loading. The test results were analyzed by using ABAQUS non linear finite element analysis. The test results obtained experimentally and those obtained through non linear finite element analysis were compared and discussed. The beam S-A1 exhibit an ultimate load of 90kN, the corresponding deflection obtained through experiment (21mm) and FEA using ABAQUS (20.55mm) varied by 2.14% and stiffness obtained through experiment (4.29 kN/mm) and FEA using ABAQUS (4.38 kN/mm) varied by 2.19 %.The beam S-B1 exhibit an ultimate load of 100kN, the corresponding deflection obtained through experiment (28mm) and FEA using ABAQUS (25.69mm) varied by 8.25% and stiffness obtained through experiment (3.57 kN/mm) and FEA using ABAQUS (3.89 kN/mm) varied by 8.99 %. The beam S-C1 exhibit an ultimate load of 115kN, the corresponding deflection obtained through experiment (35mm) and FEA using ABAQUS (33.56mm) varied by 4.11% and stiffness obtained through experiment (3.29 kN/mm) and FEA using ABAQUS (3.43 kN/mm) varied by 4.29 %. The beam S-A2 exhibit an ultimate load of 85kN, the corresponding deflection obtained through experiment (19mm) and FEA using ABAQUS (18.84mm) varied by 0.84% and stiffness obtained through experiment (4.47 kN/mm) and FEA using ABAQUS (4.51 kN/mm) varied by 0.85 %. The beam S-B2 exhibit an ultimate load of 95kN, the corresponding deflection obtained through experiment (24mm) and FEA using ABAQUS (22.66mm) varied by 5.58% and stiffness obtained through experiment (3.96 kN/mm) and FEA using ABAQUS (4.19 kN/mm) varied by 5.91 %. The beam S-C2 exhibit an ultimate load of 105kN, the corresponding deflection obtained through experiment (26mm) and FEA using ABAQUS (23.97mm) varied by 7.81% and stiffness obtained through experiment (4.04 kN/mm) and FEA using ABAQUS (4.38 kN/mm) varied by 8.47 %. Figure2 explains Load Vs Deflection behavior of ABAQUS Vs Experimental values.

Table 2. Static LoadFEA Test Results

Beam ID	**Ultimate Load**	**Ultimate Deflection**			**Stiffness**		
		Expt.	**FEA**	**Error**	**Expt.**	**FEA**	**Error**
	kN	**mm**	**mm**	**%**	**mm**	**mm**	**%**
S-A1	90	21	20.55	2.14	4.29	4.38	2.19
S-B1	100	28	25.69	8.25	3.57	3.89	8.99
S-C1	115	35	33.56	4.11	3.29	3.43	4.29
S-A2	85	19	18.84	0.84	4.47	4.51	0.85
S-B2	95	24	22.66	5.58	3.96	4.19	5.91
S-C2	105	26	23.97	7.81	4.04	4.38	8.47

	
Stress Contour of Beam S-A1	**Static Load-Deflection Response of Beam S-A1**
	
Stress Contour of Beam S-B1	**Static Load-Deflection Response of Beam S-B1**

	
Stress Contour of Beam S-C1	**Static Load Deflection Response of S-C1 Beam**
	
Stress Contour of Beam S-A2	**Static Load Deflection Response of S-A2 Beam**

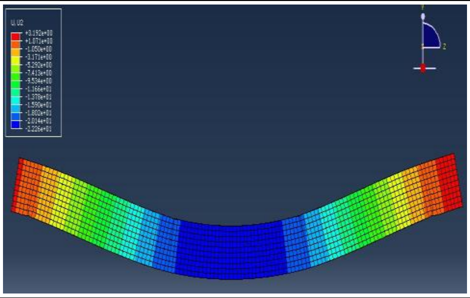
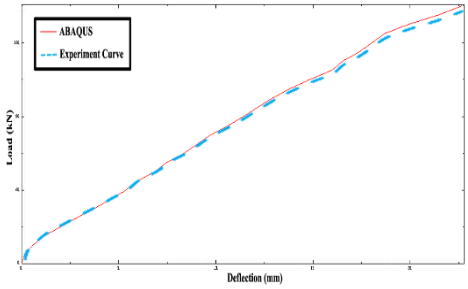
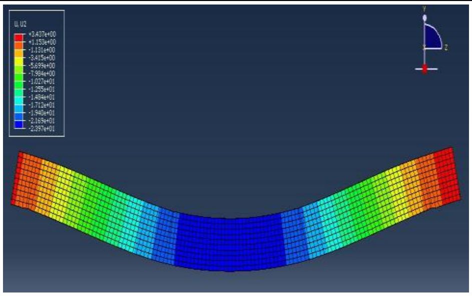
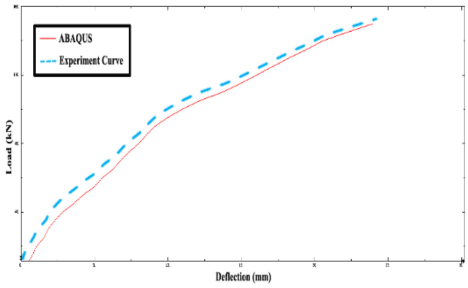
	
Stress Contour of Beam S-B2	**Static Load Deflection Response of S-B2 Beam**
	
Stress Contour of Beam S-C2	**Static Load Deflection Response of S-C2 Beam**

Figure 2. Load Vs Deflection values (ABAQUS Vs Experimental)

CONCLUSION

1. The failure Pattern of High strength ConcreteBeam satisfactory using ABAQUS and the failure load measured from ABAQUS in very close to the failure load predicted at the time of experimental works.
2. When compared to Numerical Analysis values and experimental values the deflection is quite satisfactory and as per with in the allowable limits.

REFERENCE

[1] ACI COMMITTEE 363 (1992), State-of-the-Art Report on High Strength Concrete, ACI 363R-92. American Concrete Institute, Detroit.

[2] **Ahmad, S.H., Khaloo, A.R., and Poveda, A.,** (1986), Shear Capacity of Reinforced High-Strength Concrete Beams, *ACI Journal Proceedings.* Vol. 83, No. 2, pp. 297-305.

[3] **Ashour, S.A.,** (2000), Effect of Compressive Strength and Tensile Reinforcement Ratio on Flexural Behavior of High-Strength Concrete Beams, *Engineering Structures,*Vol 22, pp. 413-423.

[4] **BAI, Z.Z., AU, F.T.K., and KWAN, A.K.H.,** (2007), Complete Non Linear Response of Reinforced Concrete Beams under Cyclic Loading, *The*

Structural Design of Tall and Special Buildings, Struct. Design Tall Spec. Build, Vol. 16, pp. 107–130.

[5] **Bhanja, S., and Sengupta, B.,** (2005), Influence of Silica Fume on the Tensile Strength of Concrete, *Cement and Concrete Research,* Vol. 35, pp. 743–747.

[6] **Cusson, D., and Paultre, P.,** (1995), Stress–Strain Model for Confined High-Strength Concrete. *Journal of Structural Engineering, ASCE*, Vol. 121, No. 3, pp. 468–477.

[7] **Ginzburg, G.Ts., Litvinova, R.E., and Borisov, A.A.,** (1976), High-Strength Concrete in Hydro Technical Construction, *UDC 691.327:666.972.52 Translated from GidrotekhnicheskoeStroitersrvo,* Vol. 3, pp. 25-26.

[8] **Hadi, M.N.S.,** (2008), Flexural Behaviour of High Strength Concrete Beams with Confining Reinforcement, ICCBT. Vol. 03, pp. 35-48.

[9] **HO, J.C.M., KWAN, A.K.H., and PAM, H.J.**, (2004) Minimum Flexural Ductility Design of High-Strength Concrete Beams. Magazine of Concrete Research, Vol. 56, No. 1, pp. 13–22.

[10] **PAM H, J., KWAN, A.K.H., and ISLAM M, S.,** (2001), Flexural Strength and Ductility of Reinforced Normal and High-Strength Concrete Beams, *Proceedings of the Institution of Civil Engineers, Structures and Buildings,* Vol. 146, No. 4, pp. 381–389.

[11] **Rajasekaran, S., Amalraj, R., and Ananda Kumar, S.,** (2000), Artificial Neural Network for Strength and Workability Prediction of High Performance Concrete Composites, *ICFRC,Proceedings of the National Seminar on High Performance Concrete composites, India*, pp. 1-18.

[12] **Ramachandra Murthy. A., Nagesh, R., Iyer, and B.K., Raghu Prasad,** (2013), Evaluation of Mechanical Properties for High Strength and Ultra High Strength Concretes, *Advances in Concrete Construction,* Vol. 1, No. 4, pp. 341-358.

[13] **Sudheer Reddy, L., Ramana Rao, N.V., Gunneswara Rao, T.D.,** (2010), Shear Resistance of High Strength Concrete Beams Without Shear Reinforcement, *International Journal of Civil and Structural Engineering,* Vol. 1, No. 1, pp. 101-113.

[14] **Suresh Reddy, R., and Ramakrishnan, K.,** (2015),Impact of Metakaolin and Silica Fume on Strength Characteristics of Concrete, *International Journal of ChemTech Research,* Vol. 8, No. 2, pp. 626-634.

[15] **Shin, S.W., Ghosh, S.K., and Moreno, J.,** (1989), Flexural Ductility of Ultra-High Strength Concrete Members. *ACI Structural Journal,* Vol. 86, No. 4, pp. 394-400.

[16] **Uhayakumar, V., Bharatkumar, B.H., Balasubramanian, K., Krishnamoorthy, T.S.,** (2007), Application of Neural Networks for Concrete Strength Prediction, *ICI Journal*, pp. 13-17.