International Journal of Technical Innovation in Modern Engineering & Science (IJTIMES)
Impact Factor: 3.45 (SJIF-2015), e-ISSN: 2455-2584
Volume 3, Issue 05, May-2017

Fracture Test & Fracture Parameters of Self Compacting Concrete using ANSYS

Zeel Vashi[1] ,Megha Thomas[2]

[1]*M. Tech Student, Parul University, Vadodara, India, zeelvashi110@yahoo.in*
[2]*Professor, Civil Engineering Department, Parul University, Vadodara, India*

Abstract - This paper presents a nonlinear finite element modeling and analysis of Self Compacting Concrete (SCC) beam subjected to three point bending load. In this study, the beam was modeled using ANSYS Workbench 17.2 nonlinear finite element software. The concrete is modeled using 'SOLID65'- which is capable of simulating the cracking and crushing behaviour of brittle materials. Grade of concrete used was M40 for the beam. The beam had an overall dimension of 430×150×100 mm. The beam is studied for the ultimate load, load-deflection and load-strain behaviour for this case and compared with the available experimental values. The above study indicates that finite element modeling is properly able to simulate the behaviour and strength of SCC beam under flexure. The Comparison study showed that the FEA predicts a 10% variation in the deflection studies.

Key Words: SCC, FEM, ANSYS, Fracture Test, Stress Intensity Factor, J Integral

I. Introduction

Self-Compacting Concrete (SCC) can be defined as a concrete that is able to flow in the interior of the formwork, filling it in a natural manner and passing through the reinforcing bars and other obstacles, flowing and consolidating under the action of its own weight (1). SCC was introduced in Japan in the late 1980's in order to overcome the congestion of steel reinforcement in case of heavily reinforced structures viz., seismic resistant structures. SCC meanwhile is spread all over the world with a steadily increasing number of applications[2]. The use of SCC offers many benefits to the construction practice: the elimination of the compaction work results in reduced costs of placement, shortening of the construction time and therefore improved productivity. Since then several attempts have been made to study the properties of SCC[3]. Concrete structures are generally analyzed either by specifically developed finite element based computer programs, or by general purpose codes that provide some kind of material model intended to be employed in the analysis of these structures. Even though the latter includes finite elements dedicated to concrete, there is no dedicated SFRC element or material law. On the other hand, although having special finite elements and material laws to represent SFRC, the specific finite element codes are in general private codes which are not always available for the research and industry communities[4].

Numerous commercial FE analysis codes are available along with the advanced modules for complex analyses. The use of FEA has increased because of progressing knowledge and capability of computer package and hardware[5]. Any attempts for engineering analyses can be done conveniently and fast using such versatile FE analysis packages. Nonlinear material models have been integrated in many of general purpose finite element codes, i.e., ABAQUS, ANSYS, STRAND7, or MSC.NASTRAN. Those nonlinear models play a vital role in nonlinear response analyses since each material component tends to possess the complicated stress-strain behavior[6]. Among those packages, ANSYS provides a three-dimensional element (SOLID65) with the nonlinear model of brittle materials similar to the concrete[7].

II. Scope of the Investigation

The finite element analysis was conducted using ANSYS, taking advantage of the wide range of element types and material models available in this computer program[8]. The analysis was performed using available material models and element formulations included in the finite element software. The study was intended to provide a better understanding of the behaviour of SCC in fracture mechanics using finite element tools[9]. The finite element models were developed, with the intent of evaluating their load-deflection behaviour and ultimate loads. Simulation results were compared with experimental data and conventional analysis theory. Since the study was performed to compare the results with already available experimental data[10].

III. Objective of the Study

- The objective of this study was to understand the flexural behaviour of SCC beam under three point bending load, and to simulate the experimental results calculated, load deformation of this type of section is agreeable or not.
- To study the behaviour of Self Compacting Concrete when loaded under the three point bending load and examine the fracture properties.

IV. FINITE ELEMENT MODELLING

The finite element method is a numerical technique of solving differential equations describing a physical phenomenon. It is a convenient way to find displacements and stresses of structures at definite physical coordinates called nodes. The structure to be analysed is discredited into finite elements connected to each other at their nodes. Elements are defined and equations are formed to express nodal forces in terms of the unknown nodal displacements, based on known material constitutive laws. Forces and initial displacements are prescribed as initial conditions and boundary conditions. A global matrix system is assembled by summing up all individual element stiffness matrices and the global vector of unknown nodal displacement values is solved for using current numerical techniques. Many software programs are available in the market for the analysis of structures by this method. In the present study, the computer program ANSYS is used for the analyses performed[11].

V. PROGRAMMING IN ANSYS

Numerical analysis of crack propagation is carried out in ANSYS, where a code have been programmed in ANSYS Parametric Design Language (APDL), determining the crack and calculating the fracture parameters like stress intensity factor and J Integral.

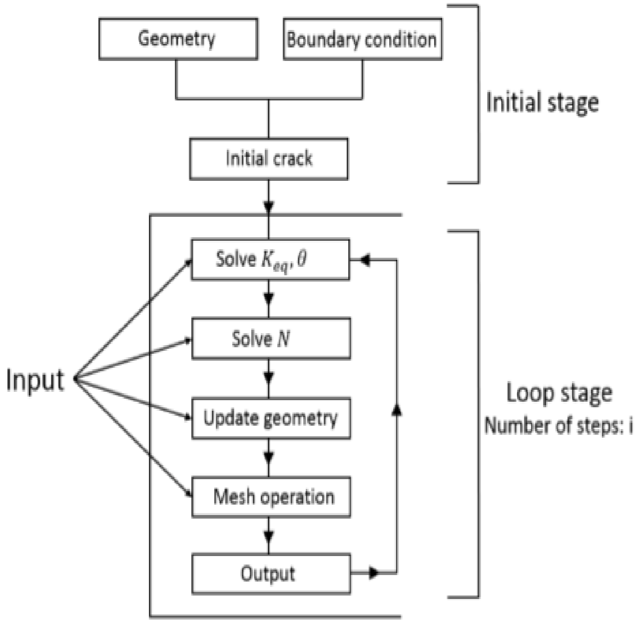


Figure 1: Diagram of APDL code in ANSYS

VI. GEOMETRY OF THE BEAM

The Main beam was made up of concrete coding and the supports and pusher of three point bending test is made up of steel member.
The beam was notched initially before testing as per the considerations as shown in Figure 2.

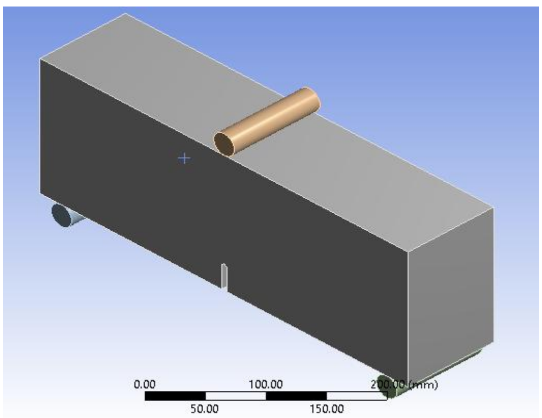


Figure 2: Geometry of Beam

VII. CONFIGURATION OF ANSYS INTERFACE

In ANSYS the interface menu from where the program is controlled during picking operations is called the GUI (Graphical User Interface), see Figure3. From this menu all the functions available in ANSYS can be selected in order to implement analyses.

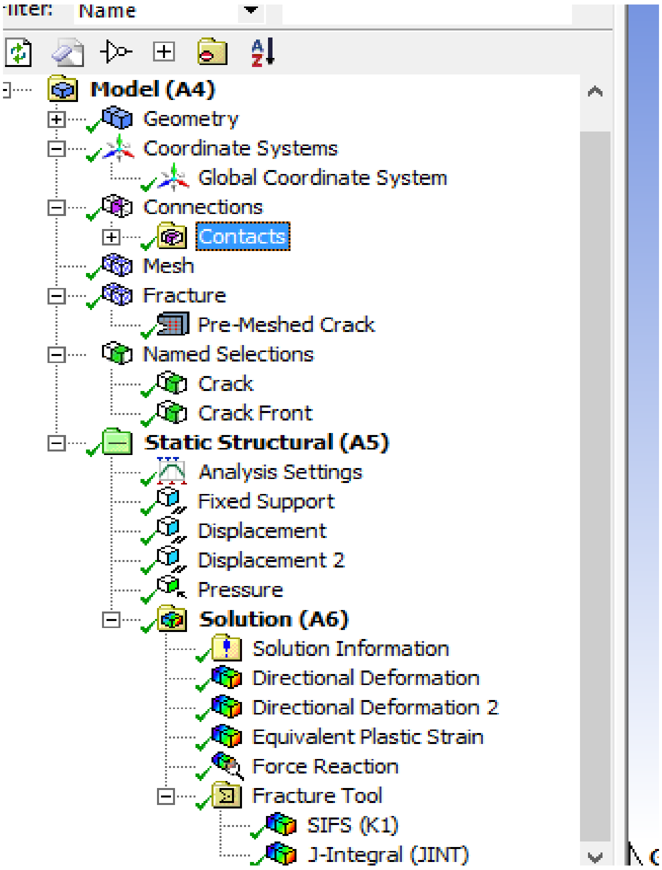


Figure 3: GUI Menu in ANSYS

VIII. MESHING

A free mesh technique can be used for meshing but it would increase both the number of elements and the computational time. Instead, the model was meshed with two objectives: to create a sufficiently fine mesh to model the essential feature of the deformed shape, and to minimize the number of elements to reduce computation time[12].
The same numbers of element divisions are considered for both concrete and steel and volumes are divided in such a format, so that the two materials share the same nodes with merging or with gluing of the volumes. However, in this study, perfect bond between materials is assumed[13].

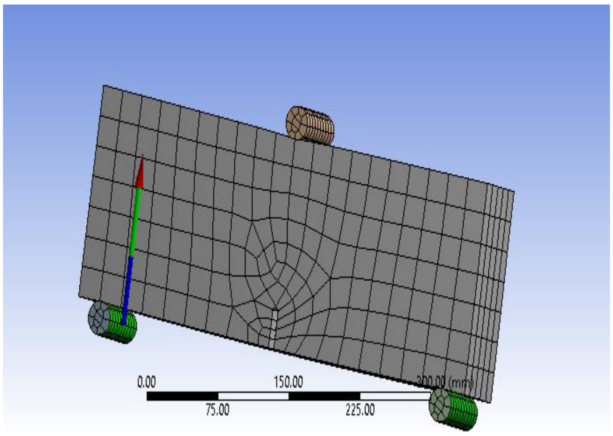


Figure 4: Meshing of Beam

IX. Boundary Conditions and Loads

Full section modeling is done of slab with appropriate boundary condition to have the better results with the experimental. The beam was tested in three point bending load. The finite element models were loaded at the same locations as the full-size beams as shown in Figure 5.

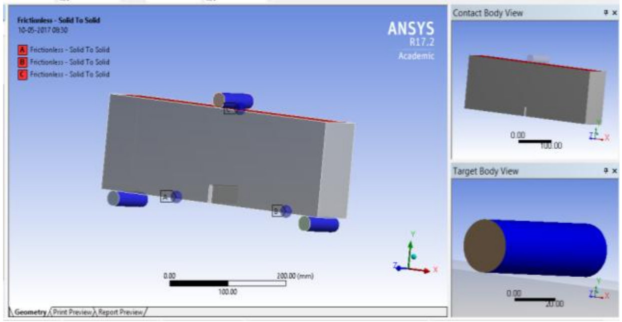


Figure 5: Boundary Conditions

X. Results

The results for displacement, stress intensity factor and J Integral are as shown in figure.
Also the graphs are shown in figure.

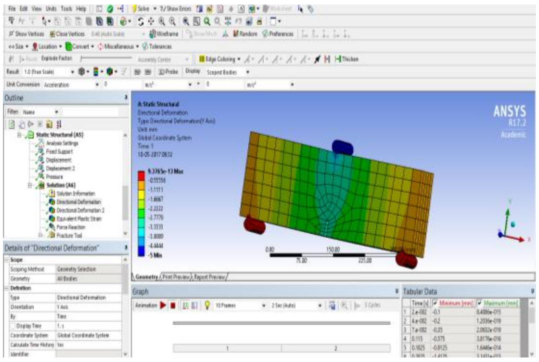


Figure 6: Results of Deformatin

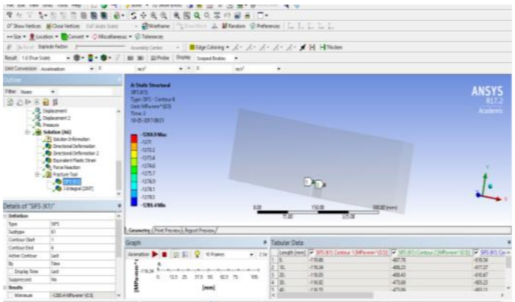


Figure 7: Results of Stress intensity factor

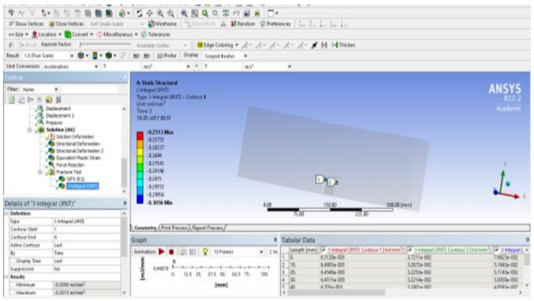
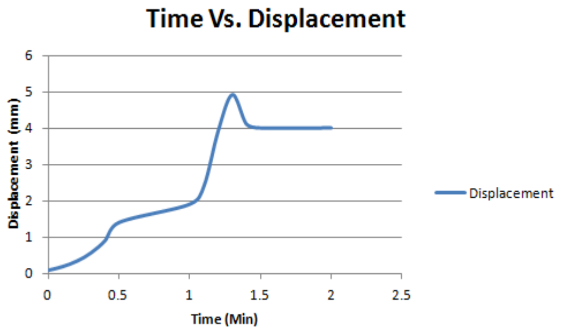
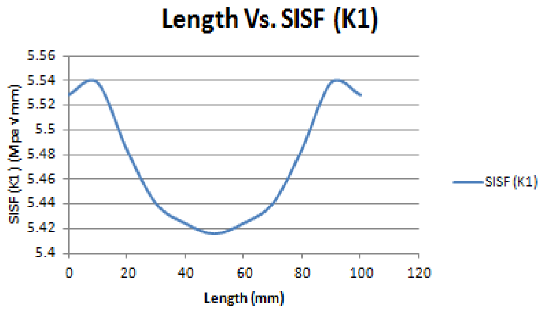
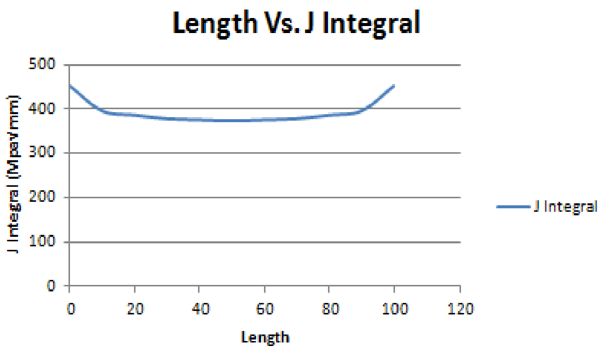


Figure 8: Results of J Integral







XI. COMPARISON WITH EXPERIMENTAL STUDIES

	Experimental	ANSYS
Stress Intensity Factor (K1) (MPa√mm)	-5.447	-4.976
J Integral (MPa√mm)	430.96	449.58

XII. DISCUSSION AND CONCLUSIONS

- The ANSYS (version 17.2) Finite Element Analysis has been used to understand the flexural behavior of M40 grade concrete beam under three point loading, and to simulate the experimental results calculated.
- The structural characteristics studied in the program are load-deflection behavior, ultimate load, ultimate stress, short term deflection and strain in the slab at working load.
- The Ultimate loads predicted by all the codes are lesser than the experimental ultimate load.
- The ratio of FEA ultimate load to experimental ultimate load being 0.98 and a variation of 12%.
- The Finite Element model predicts a 10% variation in the stress intensity factor and 12% variation in the J Integral.
- Load increment plays a significant role in the convergence of solution. Displacement convergence method proved efficient with respect to analysis time and storage space.
- The crack patterns at the final loads from the finite element models correspond well with the observed failure modes of the experimental beam

- The general conclusion is that the 3D ANSYS model is able to properly simulate the non linear behavior of the self compacting concrete beams under flexure. The general behavior of the finite element models shows good agreement with observations and data from the experimental tests.

REFERENCES

[1] T. L. Anderson, Fracture Mechanics, Fundamentals and Applications, 2005.
[2] R. I. Stephens, A. Fatemi, R. R. Stephens og H. O. Fuchs, Metal Fatigue in Engineering, 2001.
[3] C. T. Sun and Z. H. Jin, Fracture Mechanics, 2012.
[4] K. Ramesh, »Indian Institutes of Technology & Indian Institude of Science,« September 2014. [Online]. Available: http://textofvideo.nptel.iitm.ac.in/video.php?courseId=112106065.
[5]»DTD Handbook,« June 2015. [Online]. Available: http://www.afgrow.net/applications/DTDHandbook/sections/page2_2_0.aspx.
[6] Y. Murakami, Stress Intensity Factors Handbook, 1988.
[7] D.A.Russell, »Acoustics and Vibration Animations,« June 2015. [Online]. Available: http://www.acs.psu.edu/drussell/Demos/waves/wavemotion.html.
[8] E. Sharon, S. P. Gross og J. Frineberg, »Local Crack Branching as a Mechanism for Instability in Dynamic Fracture,« Physical Rewiew Letters, 1995.
[9] F. Erdogan og G. C. Sih, »On the Crack Extension in Plates Under Plane Loading and Transveres,« American Society of Mechanical Engineers, 1963.
[10] H. A. Richard, M. Fulland og M. Sander, »Theoretical Crack Path Prediction,« Blackwell Publishing, 2004.
[11] K. Tanaka, »Fatigue Crack Propagation from a Crack Inclined to the Cyclic Tensile Axis,« Engineering Fracture Mechanics, 1974.
[12] L. Liu, »Modelling of Mixed-Mode Fatigue Crack Propagation,« 2008.
[13] P. Hübner, H. Scholsser, G. Pusch og H. Biermann, »Load History Effects in Ductile Cast Iron for Wind Turbine Components,« ScinceDirect, 2007.
[14] M. J. Paris, N.-H. Kim og T. Davis, »Reanalyses of the Extended Finite Element Method for Crack Initation and Propagation,« AIAA, 2010.

