Middle-East Journal of Scientific Research 21 (9): 1663-1669, 2014
ISSN 1990-9233

DOI: 10.5829/idosi.mejsr.2014.21.09.913

Crack Chronology of Reinforced Concrete Beam under Impact Loading

[1]M.M. Hossain, [2]M.R. Karim, [3]M.A. Islam and [1]M.F.M. Zain

[1]Faculty of Engineering and Built Environment,
University Kebangsaan Malaysia (UKM), Malaysia
[2]Department of Civil Engineering,
Dhaka University of Engineering and Technology (DUET), Bangladesh
[3]Department of Civil Engineering,
Khulna University of Engineering and Technology (KUET), Bangladesh

Abstract: For many years, concrete structures have been studied to obtain the capability of endurance of impact throughout the experiments. However, in recent years, there has been an increased utilization of the multi-physic simulation software packages especially using the finite element approach. The main objective of this study is to determine the crack and spallation on reinforced concrete beam due to low velocity impact loading. Beam with geometry 1210mm (48in.) long, 200mm (8 in.) depth and 150mm (6 in.) width had been prepared. The reinforced concrete beam was impacted by a stone weight which was dropped from a speci?c height of 762mm (30 in.). The first crack occurred in the beam after 8 blows and spallation of concrete occurred after 10 blows during experiment where on finite element software (ANSYS) model the first crack occurred after 10 blows and spallation of concrete occurred after 12 blows from the top of the concrete beam. Final cracks and spallation pattern for both experimental study and ANSYS analysis is carried out for the validation of experimental analysis where the results are close.

Key words: Crack · Spallation · Reinforced concrete · Impact loading · ANSYS

INTRODUCTION

Concrete structural components exist in buildings in different forms. Understanding the response of these components during loading is crucial to the development of an overall efficient and safe structure.Generally, in structural engineering, impact loading is mostly accounted for walls, beams and slabs in both steel and concrete.The damage of impact is usually caused by projectiles, vehicle crashes, falling objects. In spite of the large number of beams designed and built, it is crucial to take impact load into consideration for the behaviour of the beams. That is the main reason that there has been a rapid growing of interest in the past few years among the engineering communities to understand the presponse of the reinforced concrete structures subjected to extreme loads due to impact and blast [1]. Although these severe transient dynamic loads are rare in occurrence for most structures, their effect can result in random and catastrophic structural failures.

Different methods have been utilized to study the response of reinforced concrete beam. Experimental based testing has been widely used as a means to analyze effects of concrete strength under loading. While this is a method that produces real life response, it is extremely time consuming and the use of materials can be quite costly [2-6]. The use of finite element analysis to study these has also been used [7-13]. However, the modelling technique still requires wide exploration and discussion in order to simulate the impact mechanism on reinforced concrete structures.The performed study investigation attempts to compare the results from elastic analysis of a reinforced beam underlow velocity impact loading, using an analysis software package (ANSYS) to that obtained

Corresponding Author: M.M. Hossain, Faculty of Engineering and Built Environment,
University Kebangsaan Malaysia (UKM), Malaysia.

from a normal Experimental analysis. This is a contribution towards a better understandingon reinforced concrete beams in terms of spallation and cracking. The prime objective of this study is to identify crack and spallation development patterns to compare the number of blows required to produce cracking andspallation between ANSYS finite element model and experimental values of reinforced concrete beam under low-velocity impact loading.

MATERIALS AND METHODS

Materials: Stone chips of Fineness Modulus (F.M) 7.46, Unit Weight 1500 kg/m^{3,} Specific Gravity 2.56 and sand of Fineness Modulus (F.M) 2.50, Unit Weight 1620 kg/m^{3,} Specific Gravity 2.60 were used to perform this experiment. Portlandcement ofUnit Weight 1120 kg/m^3 and Specific Gravity 3.15 was used.

Mix Preparation: Mix design was done to produce concrete with sufficient strength to be used. A 2500 psi target compressive strength was used to design the control mixes and the mixes were designed according to the ACI Standard Practice ACI 211.1-9114.5 [14]. Coarse aggregate of 20 mm maximum size was used in mix and slump was 50 mm. All mixtures were mixed manually without using mixer machine. Mixing procedures were the same for all of the concrete mixes. As for the concrete mixtures, the cement, coarse and fine aggregates were weighted by mix proportion and mixed constitute manually by labor. Water was then added gradually to the mix to produce a uniform mix. Standard cylinders (100mm diameter X 200mm in height) and beams (150mm X 200 mm X 1210mm) specimens were prepared for compressive strength, splitting tensile strength and flexural strength respectively.

Mechanical Properties of Concrete: This section describes details of the mechanical properties of control and the various concrete mixes. The compressive strengths of c oncrete specimens were determined after 7 and 28 days of standard curing. The load was applied in accordance with the ASTM C39 [15]. The test results are summarized in Table 1.

The modulus of elasticity is applicable within the customary working stress range (0 to 40% of ultimate concrete strength) [16].The values of modulus of elasticity (EX) and Poisson's Ratio (PRXY)are shown in Table 4.

Table 1: Compressive strength

Compressive Strength (psi)	Average Compressive Strength (psi)
7 days	1620
	1418
	1260
	1374
28 days	2944
	2652
	2387
	2626

Fig. 1: Compressive strength test

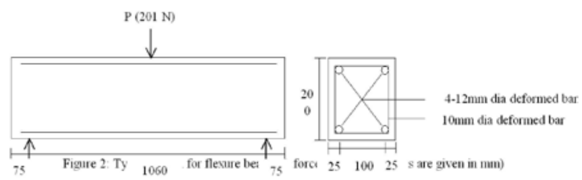

Fig. 2: Typical detail for flexure beam reinforcement (Dimensions are given in mm)

Fig. 3: Loading arrangement and supports for the beam.

Experimental Beam: The width and height of the beam tested was 150mm(in.) and 200mm (8 in.) respectively. The length of the beam was 1.21m with supports located 75mm(3 in.) from each end of the beam allowing a simply supported span of1060mm(42 in.). The mild steel longitudinal reinforcements used were 4-12mm dia deformed bars. The layout of the reinforcement is detailed in Figure 2.

The impact testing machine used in the present investigation is of pendulum type instrumented vertically impact testing as shown in Figure 3. The load which produces impact on beam was placed at the middle point of test beam. The beam was placed between two frames (Figure 4) for vertical support. Two steel bearing plates of

Fig. 4: Spallation of concrete occurs due to low velocity impact.

25mm were used under the support.The reinforced concrete beam was impacted by a stone weight, which is dropped from a speci?c height of 762mm (30 in.). The weight used in the experiment has a curved contact surface with weight of 201N. The weight was hanged by using rope and fall on to the beam after every 5 second interval. To control the movement of the beam and to make the beam as simply supported the movement of the beam was restricted by using steel plates.

The weight was hanged by using rope and fall on to the beam after every 5 second interval. Due to the low velocity impact the first crack occurs in the top after 8 blow. After 10 blows the spallation of concrete occurred.

Modeling and Analysis

Finite Element Model Using Ansys: The Finite Element Analysis (FEA) method, originally introduced by Turner *et al*. (1956) is a powerful computational technique for approximate solutions. ANSYS is engineering software, worldwide used by researchers for simulation.It develops general purpose of finite element analysis. To create the finite element model in ANSYS (SAS 2005) there are multiple tasks that have to be completed for the model to run properly. Models can be created using command prompt line input or the Graphical User Interface (GUI). For this model, the GUI was utilized to create the model. This section describes the different tasks and entries into used to create the FE calibration model.

Element Type: An element type is identified by a name, consisting of a group label and a unique identifying number. The element types for this model are shown in Table 2.

SOLID65 is used for the 3-D modeling of solids with or without reinforcing bars [17]. The solid is capable of cracking in tension and crushing in compression. In concrete applications, for example, the solid capability of the element may be used to model the concrete while the rebar capability is available for modeling reinforcement behavior. Other cases for which the element is also applicable would be reinforced composites.

Table 2: Element types for working model

Material type	ANSYS Element
Concrete	Solid 65
Steel plates and Supports	Solid 45
Steel Reinforcement	Link 8

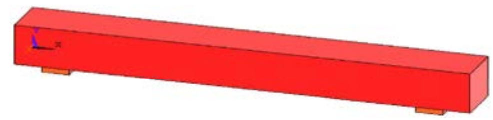

Fig. 5: Volume created in ANSYS

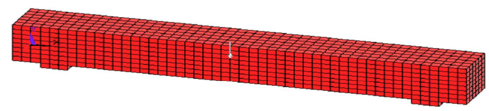

Fig. 6: Mesh of the beam model

SOLID45 is used for the 3-D modeling of solid structures [17]. In this model, SOLID45 was used for steel plates at the supports for the beam.LINK8 is a spar which may be used in a variety of engineering applications. This element can be used to model trusses, sagging cables, links, springs, etc. The 3-D spar element is a uniaxial tension-compression element with three degrees of freedom at each node. Plasticity, creep, swelling, stress stiffening and large deflection capabilities are included. LINK8 element was used to model steel reinforcement [18].

Real Constant: The real constants for this model are shown in Table 3. Note that individual elements contain different real constants. No real constant set exists for the Solid45 element. Real Constant Set 1 is used for the Solid65 element. It requires real constants for rebar assuming a smeared model. Values can be entered for Material Number, Volume Ratio and Orientation Angles. In the present study the beam is modeled using discrete reinforcement. Therefore, a value of zero was entered for all real constants which turned the smeared reinforcement capability of the Solid65.Real Constant Set 2 is defined for the Link8 element. Values for cross-sectional area and initial strain were entered. Cross sectional area refers to the reinforcement of 10mm dia deformed bar due to symmetry.

Material Properties: Material properties provided on ANSYS were on FPS unit (Table 4) for control beam.

Modeling and Meshing: The beam, plates and supports were modeled as volumes. Square mesh was set up. The overall mesh of the concrete, plate and support volumes is shown in Figure 6.The meshing of the reinforcement is

Table 3: Real Constants for Model

Real Constant	Element Type		Constant: Real constant for Rebar 1	Real constant for Rebar 2	Real constant for Rebar 3
1	SOLID65	Material Number	0	0	0
		Volume Ratio	0	0	0
		Orientation Angle	0	0	0
2	LINK8	Cross-sectional Area (in^2)	0.20		
		Initial Strain (in/in)	0		

Table 4: Material properties for the control beam

No.	Element Type		Strain (in/in)	Stress (psi)
		Material Properties		
		Linear Isotropic		
		EX		1000000 psi
		PRXY		0.15
		Multi-linear Isotropic (MacGregor Nonlinear Model)		
1	SOLID65	Point1	0.002	1300
		Point2	0.0024	1550
		Point3	0.0029	1800
		Point4	0.0034	2000
		Point5	0.004	2200
		Point6	0.0048	2400
		Point7	0.0059	2600
		Point8	0.0068	2652
		Concrete		
		ShrCf-Op		0.3
		ShrCf-Cl		1
		UnTensSt		452
		UnCompSt		2652
		BiCompSt		0
		HydroPrs		0
		BiCompSt		0
		UnTensSt		0
		TenCrFac		0
2	SOLID45	Linear Isotropic		
		EX		29000000 psi
		PRXY		0.3
3	LINK8	Bilinear Isotropic		
		Yield Stress		60000 psi
		Tang Mod		2900 psi

Table 5: Commands Used to control Nonlinear Analysis

Analysis Options	Small Displacement Transient
Calculate Pre-stress Effects	No
Time at End of Load step	500
Automatic Time Stepping	on
Number of Sub steps	2
Time increment	on
Time step size	5
Minimum time step	5
Maximum time step	6
Write Items to Results File	All Solution Items
Frequency	Write Every Sub step

Table 6: Commands Used to Control Output

Program behavior upon non convergence	Terminate but do not exit
Nodal DOF Sol'n	0
Cumulative iteration	0
Elapsed time	0
CPU time	0

Table 7: Nonlinear Control Settings Used

Equation Solver	Transient effect
Number of Restart Files	1
Frequency	Write Every Sub step

a special case compared to the volumes. The necessary mesh attributes need to be set before each section of the reinforcement is created [10].

Type of Analysis: After preprocessing, the model generation, including meshing is completed. It's ready to begin the solution phase of the ANSYS session. First, the analysis type is specified static. To restart an analysis after the initial run or load step has been completed, restart option is used. The Sol'n controls command dictates the use of a linear or non-linear solution for the finite element model. Typical commands utilized in a nonlinear transient analysis are shown in Table 5.

The commands used for the nonlinear algorithm and convergence criteria are set to defaults ANSYS (SAS 2005). The values for the convergence criteria are set to defaults except for the tolerances. The tolerances for force and displacement are set as 5 times the default values. Table 7 shows the commands used for the advanced nonlinear settings. The program behavior upon non convergence for this analysis was set such that the program will terminate but not exit. The rest of the commands were set to defaults.

To ensure that the model acts the same way as the experimental beam boundary conditions need to be applied at points of symmetry and where the supports and loadings exist. The symmetry boundary conditions were set first. The support was modeled in such a way that a roller was created.

RESULTS AND DISCUSSIONS

A calculated crack profile does not reflect the cracking history of the structure, but, rather, relates only to that particular load stage [19].In other words, the direction of cracks constantly changes and it's not possible to sketch the crack directions of the preceding load stages.For this reason, several load stages need to be examined for a complete analysis of the estimated crack pattern.Moreover, the crack condition is estimated for the concrete beam, whereas the cracks in the specimen would develop singly over a region.

Experimental Investigation of Crack Pattern: At experimentally the crack patterns were investigated by visual observation. Predominant failure will be flexural failure with some crushing beneath the impactor and some shear cracking in the impact zone [20]. Vertical cracks starting from the top of a beam were found along thebeam section. Figure 7 and Figure 8 shows the impact load–time histories, together with a number of images taken from the camera, to show thecrack development adjacent to the impact zone for the beam. Vertical cracks starting from the top of the beam were found. For the crack patterns, the first crack observed after 8 blows from the top face of the beam is shown in Figure 7and the crack enlarged after increase of number of blow. Separation between the impactor and beam occurred after 10 blows and both the beam and striker continued to move downwards. At this time some spallation of the concrete can be observedfrom the top of the beamdirectly beneath the impactor shown in Figure 8. Then the impactor and beam then regained contact, more cracking occurred and the spallation became more apparent.

Failure region is affected by inertia and the area tends to be smaller during impulsive loading. Failure region is the area where the internal energy is released and when the area is small, the cracks will be large and failure condition gets bad [21].Referring to Figure 9 it can be seen that there was local damage in the form of deep spallation on top face of the concrete beam due to low velocity impact loading. So, finally the damage indicated just below the loading spare dropped on the experimental beam.

Investigation of Crack Pattern by Using Ansys: In the non-linear region of the response,subsequent cracking occurs as more loads are applied to the beam. Cracking increases in the constant moment region and the beam starts cracking out towards the supports [22].The ANSYS program records a crack pattern at each applied load step.Figure 10 and 12 shows the mechanism of damagewave propagation from the initial potential fracturingregion under the zone of impact towards the support. Figure 10 also shows the development of cracks (damage line indicator) closer to the centre of the beam.The initial cracking of the concrete beam in the ANSYS model corresponds at sub steps 10 from the top of theconcrete beam and propagating from the projectile towards thesupport shown in Figure 11. The reason of it is the energy wave propagates tothe support and then reflected.

Fig. 7: First crack occurs after 8 blows

Fig. 8: First spallation occurs after 10 blows.

Fig. 9: Deep spallationoccurs after 32 blows.

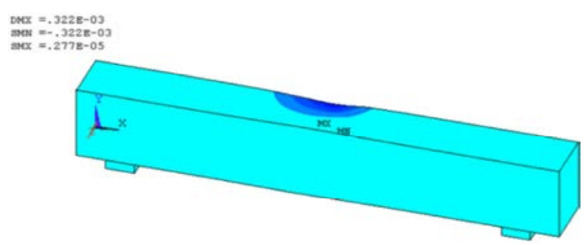

Fig. 10: Spallation and deflection at sub step 10

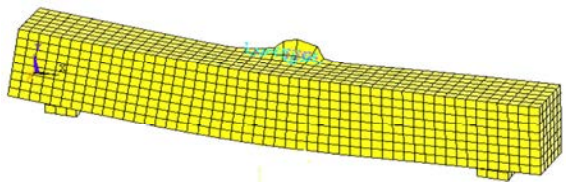

Fig. 11: Crack investigation at sub step 10

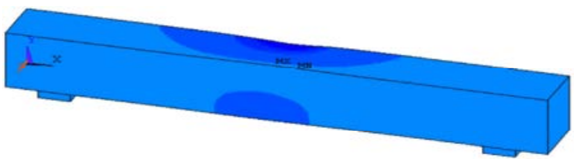

Fig. 12: Propagation of spallation at sub step 11

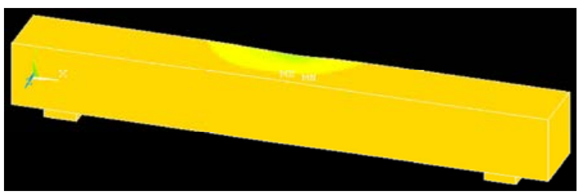

Fig. 13: Propagation of spallation at sub step 12

Table 8: Comparison between results

Events	From Experiment	From ANSYS
No. of blows to observe first crack	8	10
No. of blows to observe Spallation	10	12

The appearance of the cracks reflects the failure modes for the beam. At sub step 13, the beam can no longer support additional load as indicated by an insurmountable convergence failure. Severe cracking throughout the entire constant moment region occurs (Figure 12) and this existing crack continues to propagate towards thesupport including the area inthe top centre of the beam (Figure 13). Before the collapse few splitting cracks appear at the upper part of the beam due to crushing failure of the concrete there. It also can be seen that the cracking and spallation in Figure 13 is denser than the Figure in 12.

Both in ANSYS and Experimental result it was shown that the crack pattern and spallation developed was proportional to time and blows. Table 8 shows the summarized forms of comparison of results obtained from experimental analysis and ANSYS finite element analysis.

CONCLUSIONS

Following conclusions can be drawn from the present study:

- The load under low velocity of repeated impact loading is approximately spherical in shape on the beam. The duration of the impact pulse remains nearly constant.
- Measured cracking patterns are approximately same as obtained from the ANSYS analysis and experimental results.
- The measured times for spallationisfound closer for the ANSYS analysis and experimental results.
- A good correlation has been observed between number of blows as obtained from the ANSYS analysis and experimental resultsboth for cracking and spallation occurring.

ACKNOWLEDGEMENTS

Foremost, authors would like to express deep sincere thanks and gratitude to all the concerned teachers and stuffs of the Civil Engineering Department of Khulna university of Engineering and Technology for their direct and indirect assistance at different stages of the work.

REFERENCES

1. Tuan Ngo and Priyan Mendis, 2008. “Modelling reinforced concrete structures subjected to impulsive loading using concrete lattice model,” Electronic Journal of Structural Engineering, pp: 8.
2. Kishi, N., H. Mikami, K.G. Matsuoka and T. Ando, 2002. “Impact Behavior of Shear-Failure-Type RC Beams without Shear Rebar,”International Journal of Impact Engineering, 27: 955-968.
3. Rokugo, Keitetsu, Minoru Kunieda, Hiroshi Sakai and Masahiro Suzuki, 2002. ‘Study on impact resistance of PC beams with buffer layer made of ductile fiber reinforced cementitious composites", In Proc. 1st Feb Congress, Osaka, Japan: Federation Internationale de Beton Feb, 2002.
4. Faham, T., 2008. Numerical modelling of reinforced concrete slabs subjected to impact loading. Master ofEngineering (Research), Faculty of Civil Engineering, University of Wollongong, Australia. pp: 48-78. N.
5. Erki, M.A. and U. Meier, 1999. “Impact loading of concrete beams externally strengthened with CFRP laminates”, Journal of Composites for Construction, pp: 3645-3659 (August 1999).

6. Buckhouse, E.R., 1997. “External Flexural Reinforcement of Existing Reinforced Concrete Beams Using Bolted Steel Channels,” Master’s Thesis, Marquette University, Milwaukee, Wisconsin.
7. Saifullah, I., M.A. Hossain, S.M.K. Uddin, M.R.A. Khan and M.A. Amin, “Nonlinear Analysis of RC Beam for Different Shear Reinforcement Patterns by Finite Element Analysis”, International Journal of Civil & Environmental Engineering IJCEE-IJENS Vol: 11 No: 01.
8. Saifullah, I., M. Nasir-uz-zaman, S.M.K. Uddin, M.A. Hossain and M.H. Rashid, “ Experimental and Analytical Investigation of Flexural Behavior of Reinforced Concrete Beam”, International Journal of Civil & Environmental Engineering IJCEE-IJENS Vol: 11 No: 01.
9. Anthony, J. and B.S. Wolanski, 2004. “ Flexural behavior of reinforced and prestressed concrete beams using Finite Element Analysis”, Master’s thesis, Marquette University, Milwaukee, Wisconsin May, 2004.
10. Kabir, M.Z. and E. Shafei, 2009. “Analytical and Numerical Study of FRP Retorted RC Beams Under Low Velocity Impact”, Journal of Composites for Construction, pp: 415-428 (October 2009).
11. Josko Ozbolt and Akanshu Sharma, 2001. “Numerical simulation of reinforced concrete beams with different shear reinforcements under dynamic impact loads”, International Journal of Impact Engineering, pp: 940-950 (December 2011).
12. Hua Jiang, Xiaowo Wang and Shuanhai He, 2012. “Numerical simulation of impact test on reinforced concrete beams”, International Journal of Impact Engineering, pp: 111-120 (August 2012).
13. Demetrios, M. Cotsovos, 2010. “A simplifiedapproach for assessing the load carrying capacity of reinforced concrete beams under concentrated load applied at high rates”, International Journal of Impact Engineering, pp: 907-917 (August 2010).
14. ACI 211.1-91 (R2009), “Standard practice for selecting proportions for Normal Heavyweight and Mass concrete”.
15. ASTM C39 / C39M - 14, “Standard Test Method for Compressive Strength of Cylindrical Concrete Specimens”.
16. ASTM C469-94, “Standard Test Method for Static Modulus of Elasticity and Poisson’s Ratio of Concrete in Compression”.
17. ANSYS 2005, Documentation for ANSYS, version 10.0
18. ANSYS workbench tutorial.
19. SelcukSaatci and Frank J. Vecchio, “Effects of Shear Mechanisms on Impact Behavior of Reinforced Concrete Beams,” ACI Structural Journal, Title no. 106-S09.
20. Chen, Y. and I.M. May, 2009. "Reinforced concrete members under drop-weight impacts." Proceedings of the Institution of Civil Engineers. Structures and Buildings 162(1): 45-56.
21. Ayaho Miyamoto, Michael E. King and Manabu Fujii, 1989. “Non-linear dynamic analysis and design concepts for RC beam under impulsive loads”, Bulletin of the New Zealand national society for earthquake Engineering, Vol. 22, No. 2, June 1989.
22. Dahmani, L., A. Khennane and S. Kaci, 2010. "Crack identification in reinforced concrete beams using ANSYS software", Strength of Materials 42(2): 232-240.