

Computational Analysis of Reinforced Concrete Slabs Subjected to Impact Loads

Shahrul Niza Mokhtar¹, Redzuan Abdullah²

¹Department of Structures and Materials Engineering, Faculty of Civil and Environmental Engineering, Universiti Tun Hussein Onn Malaysia, 86400, MALAYSIA.

²Department of Structures and Materials, Faculty of Civil Engineering, University Technology of Malaysia, MALAYSIA

Abstract: Nowadays, the numerical models for the impact load assessment are starting to become more accurate and reliable. Combined with modern computer hardware, the computational time for such an assessment has been reduced to a satisfactory level. In this study, an attempt has been made to present the simulation technique and examine the accuracy of modern software with regards to assessing the response of reinforced concrete slabs subjected to impact loading near the ultimate load ranges. The response such as time-impact force graph, damage wave propagation, effectiveness of mesh density, effect of projectile size and final crack pattern are verified against existing experimental results. It is shown that the present general purpose Finite Element Analysis (FEA) is able to simulate and predict the impact behavior of structural systems satisfactorily.

Keywords: Computational Simulation, Reinforced Concrete Slabs, ABAQUS, Impact Loads

1. Introduction

In the last decade, investigation in structural engineering has progressively more considered on behavior of the reinforced concrete (RC) element further than the elastic range and situation where dynamic response is encountered such as by Saatci et al, Abbas et al and Nazem et al. Structural elements might initiate failure when expose to various extreme loading conditions during their serviceability process. Impact loading is the one of the important loading types that a structural element may have to sustain.

RC structures are often subjected to extreme dynamic loading conditions due to direct impact. Typical examples include transportation structures subjected to vehicle crash impact, marine and offshore structures exposed to ice impact, protective structures subjected to projectile or aircraft impact, and structures sustaining shock and impact loads during explosions [4]. Understanding the structural behavior especially slabs element to impact load is essential to protect this critical members from collapse and fail. Moreover, in order to ascertain a reliable impact-resistant design procedure of slabs elements, a series of practical tests are required.

Estimating the response of RC structures to impact loading through full-scale tests is expensive in terms of providing the necessary test material, test equipment, and time to perform. Many researchers such as [5] – [7], have successfully investigated the impact failure of RC elements by practical tests. However, the modeling technique still requires wide exploration and discussion in order to simulate the impact mechanism on RC structures. Thus, this paper describes the numerical modelling

technique and investigations into the response of an RC slabs as well as the steel reinforcement failure mechanism when subjected to impact loading in aspects of failure. In order to gain the better understanding of the behavior of the structure, the Finite Element (FE) analysis has been carried out using ABAQUS software by utilizing different non-linear material models which are available in the ABAQUS/Explicit material library. The numerical results are further discussed by validating with experimental.

2. Experimental Work

Practical test were carried out at Heriot-Watt University in Edinburgh by Chen et al investigating high mass – low velocity impact behavior of reinforced concrete slab and the resulting dynamic response of the total structure. Tests were carried out on several concrete slabs with grade 40 under drop-weight loads as shown in Figure 1.

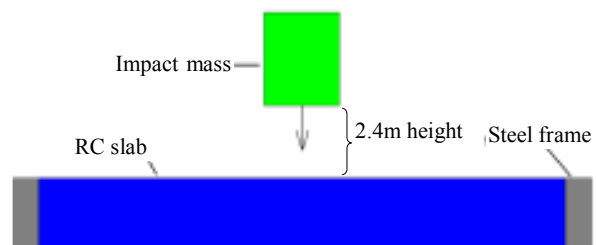


Figure 1: Schematic diagram of the RC slab experiment setup

The total size of the slab is 760 mm x 760 mm in length and width, 76 mm in depth. The size of the concrete region is 725 mm x 725 mm in length and width, 76 mm in depth. The slab is reinforced with 6 mm diameter high yield steel bars as top and bottom reinforcement. The concrete cover between the main reinforcement bars and the top and bottom edges of the slab is 12 mm. The main reinforcement bars are spaced at 60 mm intervals. In addition, a cylindrical impact mass is used. The diameter of it is 100 mm, and the weight of it is 98 kg. The steel drop-weight is acted vertically from a certain height 2.15 m (correspond to the impact speed 6.5 m/s). The outputs of the test set-up (load cell, accelerometers, strain gauges and electronic triggers) were amplified and then fed in a data logger that can operate at rates of up to 50 MHz. The details of steel reinforcement arrangement and the dimension of projectile are shown in Figure 2.

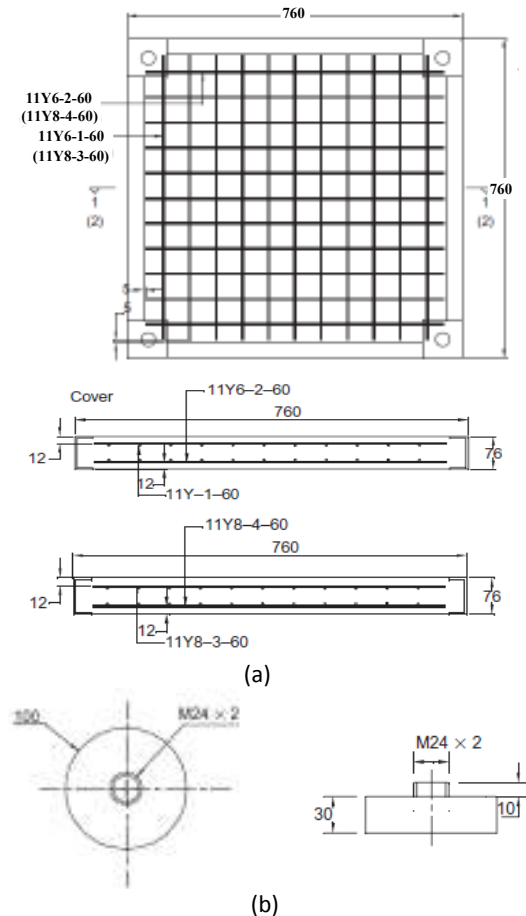


Figure 2: The details of the (a) RC slabs and (b) steel projectile

3.0 Computational non-linear simulations

The simulation of finite element models of reinforced concrete slabs were developed by using three dimensional solid elements. The modelling process including discretized geometry, element section properties, material

data, loads and boundary conditions, analysis type, and output requests were addressed.

Element's modelling

Firstly, the eight-node continuum elements (C3D8R) for slabs with three different materials were created. Secondly, the steel reinforcement was modeled by two-node beam elements connected to the nodes of adjacent solid elements. In addition, 6-mm diameters for top and bottom reinforcement were developed. For the steel support of an RC slab model, discrete rigid element was developed. Finally, the impact load (steel projectile) was developed by continuum solid and revolved to 360° for produce the cylindrical shape.

Element's interaction

The individual modelled elements should be connected properly to each other after assembling the structural and non-structural elements. Tie contact technique was utilized to create proper interaction between un-deformable discrete rigid element (steel support) and solid element (concrete slab) as shown in Figure 3. This technique can avoid the shear interaction between these two elements. In this investigation, the embedded technique was used to constraint the two-node beam elements (steel reinforcement) into solid element (concrete slab) in order to create a proper bond action. Surface-to-surface contact (explicit) is defined for interaction between the impact load (steel projectile) and solid element (concrete slab). Furthermore, the kinematic contact method for mechanical constraint formulation was employed in defining the contact property option. In this simulation, a friction coefficient of 0.2 is used for all contact surfaces.

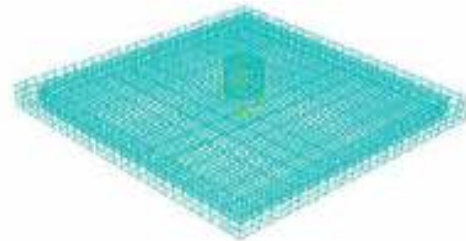


Figure 3: Render model of simulation (wireframe)

Constitutive model of concrete

For non-linear FE analysis, material model can play an essential role in order to predict the strength of concrete. In this study, the model is consists of two behaviors, which are ductile model and brittle-cracking model. Therefore, three different types of material behavior such as linear pressure dependent (i) Drucker-Prager (DP) model (ii) Cap-Plasticity (CP) model are characterized as ductile model, and meanwhile, (iii) Concrete Damage Plasticity (CDP) model is represented for brittle-cracking model. These models have been addressed to enhance the understanding of visco-elastic,

visco-plastic and post-cracking in RC structures subjected to impact loading.

For DP model, the constitutive equation is written in equation (1) and can be illustrated in Figure 4;

$$f_{DP} = \sqrt{J_{2D}} - \alpha I_1 - k \quad (1)$$

where J_{2D} is a second invariant of the deviatoric tensor, while α and k is positive material constant which α is defined as dilation angle.

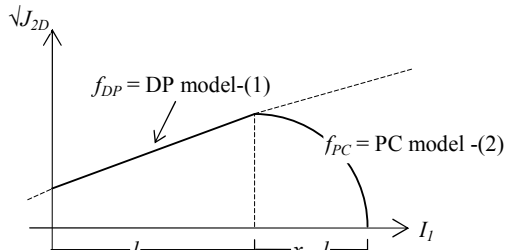


Figure 4: DP and PC surface on I_1 and $\sqrt{J_{2D}}$ plot

In order to get the better and realistic simulation performance, the parameters are considered from conventional properties of normal strength of concrete, in which, some of the data taken from previous work by [6], example from [8]-[9] and also individual experimental tests data by [5] were used.

Table 1 indicates the details of the parameters used in this analysis, while Table 2 shows the sub-option of DP hardening parameters during plasticity takes place.

Table 1: Drucker-Prager parameters

Angle of friction	Flow stress ratio	Dilation angle
30^0	1	20^0

Table 2: Drucker-Prager hardening parameters

Yield Stress (Pa)	Abs Plastic Strain
13000000	0
20000000	0.0007
24000000	0.001
37500000	0.002
22500000	0.0034
16000000	0.05

The parameter in Table 2 corresponds to the Figure 5 in order to illustrate the plasticity behavior of concrete when utilizing DP model.

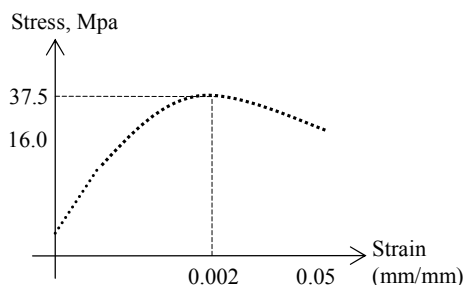


Figure 5: Stress-strain relationships for DP model

For CP model, the elliptic strain-hardening cap model is utilized to control the plastic volumetric change. The constitutive equation is shown in (2) and plotted as in Figure 4.

$$f_{PC} = (I_1 - l)^2 + R^2 J_{2D} - (x - l)^2 \quad (2)$$

where l is denoted as initial cap yield surface and R is ratio of major to minor axis of elliptic cap, which may be a function of l . The parameters used in this model are shown in Table 3.

Table 3: Cap Plasticity concrete model parameters

Material Cohesion (Pa)	Material Angle of Friction (β)	Cap Eccentricity Parameter (R)
4705672	51	0.65
Initial Cap Yield Surface Position	Flow Stress Ratio	Strain Rate Effect
0.0011	1	1.5

Finally, the CDP model is employed to predict the impact behavior of RC slabs. In this model, the constitutive parameters are properly studied in order to simulate the reliable response of structural system. The function of this model is expressed in equation (3) and (4).

$$f_{CDP} = \omega \left[\sqrt{3J_{2D}} + \alpha I_1 + \beta \langle \sigma_{max} \rangle - \gamma \langle -\sigma_{max} \rangle \right] \quad (3)$$

with

$$\omega = \frac{1}{1 - \alpha} \quad (4)$$

where β and γ are dimensionless constants. Further explanation regarding CDP model can be obtained in study of [8].

Table 4 shows the constitutive parameters used in this model for both tension and compression region.

Table 4 Concrete Damage Plasticity model parameters

Plasticity				
Main option				
Dilation Angle	Eccentricity	f_{bo}/f_{co}	K	Viscosity Parameter
38^0	1	1.12	1	0.666
Compression behavior				
Main-option			Sub-option	
Yield Stress (MPa)	Inelastic Strain	Rate	Damage Parameter	Inelastic Strain
13.0	0	1.5	0	0
20.0	0.0007	1.5	0	7.473×10^{-5}
24.0	0.001	1.5	0	9.885×10^{-5}
37.5	0.002	1.5	0	0.0001541
22.5	0.0034	1.5	0	0.0007615
16.0	0.05	1.5	0.195402	0.0025576

			0.596382	0.0056754
			0.894865	0.0117331
Tension behavior				
Main-option			Sub-option	
Yield Stress (MPa)	Cracking Strain	Rate	Damage Parameter	Inelastic Strain
3.500	0	1.5	0	0
1.750	0.00015	1.5	0	3.333x10 ⁻⁵
0.800	0.00035	1.5	0.406411	0.0001604
0.250	0.0006	1.5	0.69638	0.0002798

As shown in the Table 4, rate of 1.5 is used as the effectiveness of the strain rate from impact loads, in which to model the increasing of compressive and tensile strength due to the short period action.

In this non-linear simulation, the material properties for concrete and steel reinforcement as well as steel projectile are shown in Table 5 and 6, respectively.

Table 5 Concrete materials properties

Young's Modulus (N/m ²)	Poisson's Ratio	Density (kg/m ³)
3.00x10 ⁺¹⁰	0.2	2400
Fracture Energy (N/m)	Concrete f _{cu} (N/mm ²)	Concrete f _{ct} (N/mm ²)
100	53	2.1

Note: f_{cu} is the concrete compressive strength.
f_{ct} is the concrete tensile strength.

Table 6 Steel reinforcement and steel projectile materials properties

Young's Modulus (N/m ²)	Poisson's Ratio	Density (kg/m ³)
2.1x10 ⁺¹¹	0.29	7800
Yield stress (N/mm ²)	Ultimate stress (N/mm ²)	
5.60x10 ⁺⁸	6.3x10 ⁺⁸	

In order to simulate the behavior of steel reinforcement, the elastic-plastic hardening behaviors are utilized in this study. See Table 7 for the preferred parameter.

Table 7 Elastic-plastic behavior for impact mass (steel projectile) parameters

Young's Modulus (N/m ²)	Poisson's Ratio	Density (kg/m ³)
2.1x10 ⁺¹¹	0.29	7800
Plasticity		
Yield stress (N/mm ²)	Plastic strain	
3.046x10 ⁺⁸	0	
3.4419 x10 ⁺⁸	0.0244	
3.8551 x10 ⁺⁸	0.0951	
4.5039x10 ⁺⁸	0.1384	

4.7028 x10 ⁺⁸	0.191
5.00 x10 ⁺⁸	0.2324
5.80 x10 ⁺⁸	0.2728

Predefined field

To simulate the motion of the projectile (drop weight), each of the nodes are given an initial velocity (6.5 m/s) in a direction perpendicular to the slabs. Therefore, the projectile struck the slab at a constant velocity of 6.5 m/s. This velocity value is given as in experimental works.

Mesh generation

In the first three-dimensional model, there are 11520 linear tetrahedral elements of type C3D8R and 14406 nodes are used to represent the concrete; 2640 linear line beam elements of type B31 with 2541 nodes to represent the reinforcement bars; 820 linear quadrilateral elements of type R3D4 and 4 linear triangular elements of type R3D3 with 822 nodes to represent the un-deformable (rigid body) steel frame; and 413 total number of linear hexahedral elements of type C3D8R with 576 number of nodes to represent the steel projectile. Furthermore, in this investigation, the Hourglass control and distortion mesh control techniques were utilized.

Output request

The critical output parameters from the numerical simulation should be compared with the experimental results, therefore, the field output request parameters were defined such as deflection, stress, strain and contact force versus time.

4.0 Results and Discussion

Impact force and rebar failure point

The first stage of this dynamic numerical analysis is to determine the time-impact force graph of each model. Then, the numerical graph pattern result is validated with the experimental results. In Figure 6, it can be seen the maximum impact force for experiment results is 140 kN. This figure also shows that impact force curve of these three models give similar pattern as compared to practical tests. However, the ductile behavior (PC) can simulate the behavior of dynamic loading in reinforced concrete structures as closely as an experiment, where, the value of impact force of PC model is approximately 130 kN.

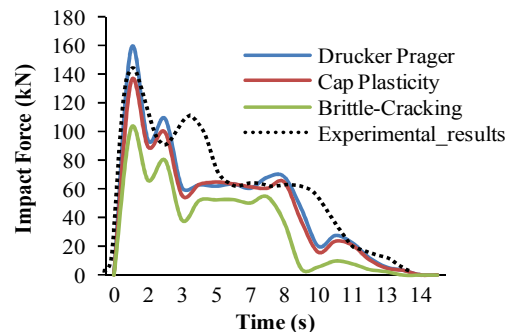


Figure 6: Time-impact force graph

To validate the failure point of rebar between numerical simulations and the experiment result, the comparisons were investigated through these two graph as shown in Figure 7. The higher stress value for steel reinforcement that obtained from numerical simulation as shown in Figure 7 (a), gives comparable failure region to the practical work in Figure 7 (b).

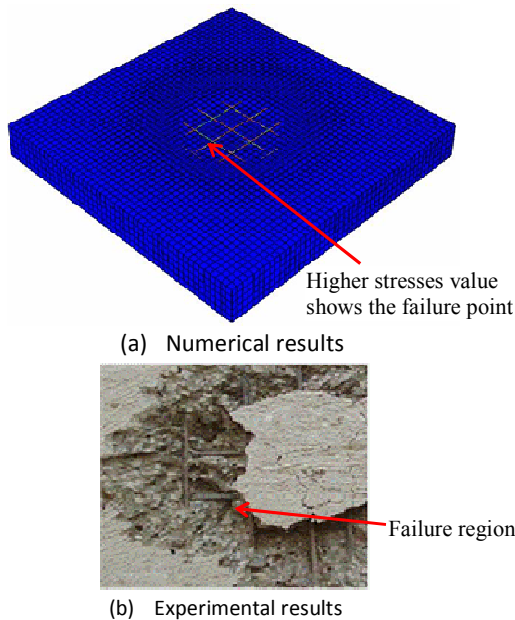


Figure 7: Failure point for the steel reinforcement

Effectiveness of mesh density

For every FE simulation, the whole domain is discretized into elements and some assumptions are made. In order to provide a reasonably accurate result for FE analysis, sufficiently refined element meshes are used. To achieve this requirement, refining the mesh can be applied by changing the number of seeds (element per edge) in the developed models. The results of the refining mesh of the slab can then be compared with the results obtained by the initial mesh.

Table 8 Number of element and nodes for each case (for Slab meshes)

Mesh type	Number of elements	Number of nodes
Coarser Mesh	100	242
Original Mesh	11520	14406
Finer Mesh	41472	47961

Table 8 shows the number of element for each case. According to the Figure 8, the values of impact force for the finer mesh of Drucker- Prager or Cap/Plasticity are in

reasonable agreement with experimental results rather than Brittle-Cracking. In addition, by using finer mesh of ductile model, the maximum value of impact force is similar to the practical results. Therefore, in this analysis, models using ductile behavior can give more realistic results than the Brittle-Cracking model. However, refining the mesh elements can increase the computational costs (time, available powerful computer processor, etc).

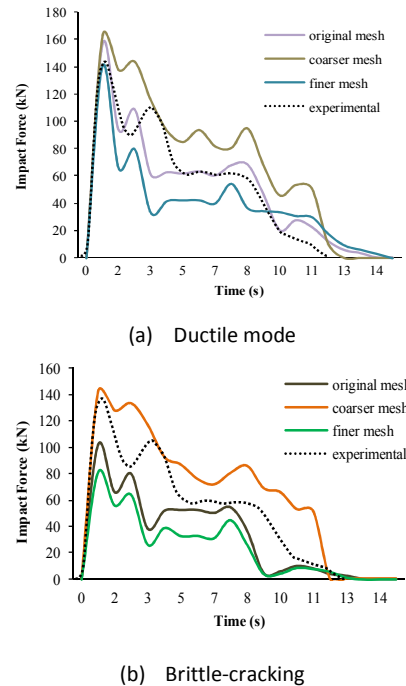


Figure 8: Time-Impact Force Graph with different mesh densities (Ductile Model) and (Brittle-Cracking)

Effect of projectile size

Figure 9 (a) shows the development of cracks (damage line indicator) closer to the centre of the slab. Furthermore, the region of the cracking area for the Figure 9 (a) is larger than the one in Figure 9 (b). It also can be seen that the cracking in Figure 9 (a) is denser than the one in Figure 9 (b). There is some new cracking developed on the top face of the slab. The reason of it is: with the same mass, the smaller size of projectile will produce higher pressure. That will result in more cracks in the vicinity of the impact. The damage area indicated by red color of the impacted slab by half projectile is quite small as compared to the original diameter of projectile. There is another thing that can be mentioned: because the size of projectile was reduced, the areas of damages were affected by the size of the projectile. However, it does still produce the higher pressure to the slab.

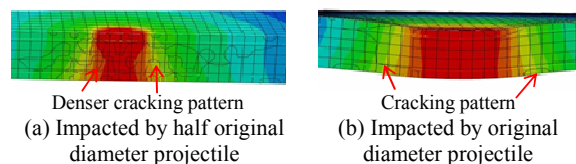


Figure 9: Numerical simulation result by using different size of projectile

Damage wave propagation

The Concrete Damage Plasticity or (Brittle-Cracking) models were utilized in order to obtain the realistic wave propagation in the slab models. Figure 10 (a), (b), (c), (d) and (e) show the mechanism of damage wave propagation from the initial potential fracturing region under the zone of impact towards the support. In Figure 10 (d), the first crack appears at the bottom of the slab and propagating from the projectile towards the support. The reason of it is the energy wave propagates to the support and then reflected. Then, Figure 10 (e) shows the existing crack continues to propagate towards the support and covers the whole slab including the area in the top centre of the slab.

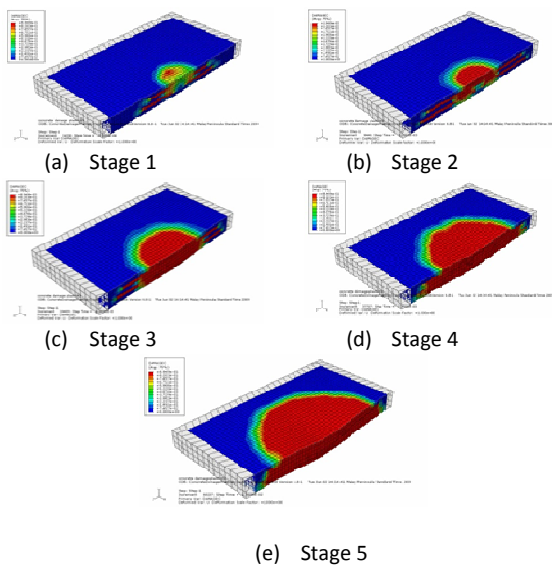


Figure 10: Damage wave propagation between 0s to 0.015s

Final crack pattern

The final crack pattern (bottom face) obtained by the experimental work is shown in Figure 11 (a). It can be seen from the test results, the shape of the cracking region is a ring: the outer diameter of the region is approximately 400 mm, and the inner diameter is approximately 180 mm. The final crack pattern (bottom face) obtained by Brittle-Cracking model simulation is shown in Figure 11 (b). This pattern corresponds well to the experimental one. Although the inner diameter of the cracking zone is smaller than the experimental, the outer diameter of the cracking zone is almost the same as the one obtained in experimental work which is approximately 400 mm. The final crack patterns (top face) obtained by both experimental work and numerical analysis are shown in Figure 11 (c) and (d). It can be seen from the figures, there only has very small cracking in the region of compressive stresses. That does not correspond to the results obtained by experiment. In the other words,

there is no effects of spallation could be observed in ABAQUS results.

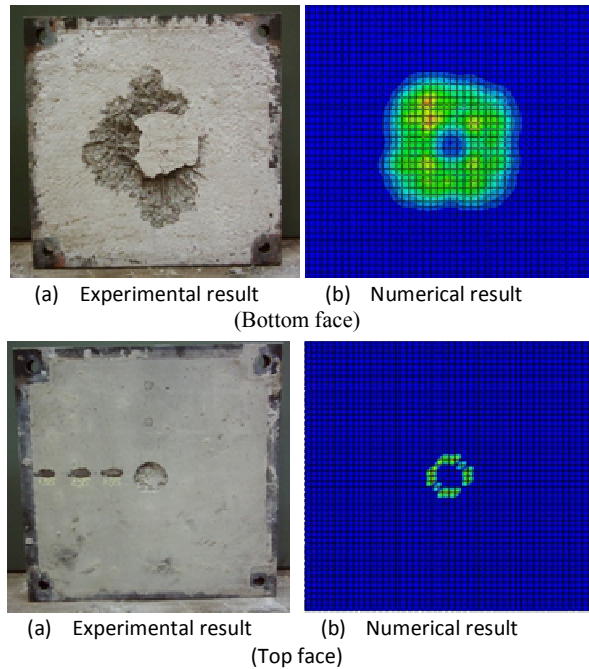


Figure 11: Crushing and final crack pattern of the impacted reinforced concrete slab

[1] Conclusion

According to the modelling result as explained in the above topics, the numerical simulation by using ABAQUS software could produce the result as closed as an experimental result. The non-linear material models which are available in the ABAQUS/Explicit material library such as Drucker-Prager and Cap-Plasticity that represent Ductile behavior give better and realistic results than the Brittle-Cracking model (Damage Concrete Plasticity). Furthermore, finite element analysis by using ABAQUS software is capable of developing reasonable and realistic estimations available in order to investigate the possible damage modes of reinforced concrete slabs under impact loads.

References

- [1] Saatci, S., and Vecchio F. J. Nonlinear finite element modeling of reinforced concrete structures under impact loads. *ACI Structural Journal*, Vol. 106, (2009), pp. 717-725.
- [2] Abbas, H., Gupta, N. K., and Alam, M. Nonlinear response of concrete beams and plates under impact loading. *International Journal of Impact Engineering*, Volume 30, (2004), pp. 1039 – 1053.
- [3] Nazem, M., Rahmani, I., and Rezaee-Pajand, M. Non-linear FE analysis of reinforced concrete structures

- using a Tresca-type yield surface. *Transaction A: Civil Engineering*, Volume 16, No. 6, pp.512 - 519.
- [4] Zineddin, M., and Krauthammer, T. Dynamic response and behaviour of reinforced concrete slabs under impact loading. *International Journal of Impact Engineering*, Volume 34, (2007), pp.1517-1534.
- [5] Chen, Y., and May, I. M., Reinforced concrete members under drop-weight impacts. *Proceeding of the Institution of Civil Engineers*, 162(SBI): (2009) pp.45-56.
- [6] Faham, T. Numerical modelling of reinforced concrete slabs subjected to impact loading. Master of Engineering (Research), *Faculty of Civil Engineering, University of Wollongong, Australia*. (2008). pp. 48-78.N.
- [7] Kishi, H. Mikami, K. G. Matsuoka and T. Ando, Impact Behavior of Shear-Failure-Type RC Beams without Shear Rebar. *International Journal of Impact Engineering*. Vol. 27. 955-968, 2002.
- [8] ABAQUS Analysis User's Manual Online Documentation. (Version 6.8), *Dassault Systèmes Simulia Corp., Providence, RI, USA*. (2008).
- [9] Lubliner, J. A plastic-damage model for concrete. *International Journal Solids Structures*. Volume 25, No. 3 (1989). Pp. 299-326.