Nonlinear dynamic response of RC building façade panels to impact loads

M. Abdel-Mooty & S. Shaaban
Department of Construction and Architectural Engineering, The American University in Cairo, Egypt

Abstract

The performance of reinforced concrete facade components such as beams or one-way slab panels to impact loads is examined in this paper. A building façade is the first line of defence against external shock and impact loads. The response of the building façade elements to such extreme loadings also influences the load transmitted to the main structural system of the building. This significantly affects the structural overall stability and integrity as well as the safety of the occupants. Nonlinear numerical simulation of the impact response of reinforced concrete beam is presented. The numerical model is developed using ANSYS/ LS-DYNA numerical code. The developed model is verified against the experimental results of a previous study carried out at AUC. The developed model is used to evaluate the effect of different factors such as mesh size, elements type, material models, and contact interface elements on the accuracy of the results. The effect of beam reinforcement, stiffness, damage, and the supporting element on the beam capacity to sustain impact load is examined. The effect of the studied parameters on the reaction forces transmitted to the supporting structure is also evaluated.
Keywords: nonlinear behaviour, numerical modelling, impact load, RC panels.

1 Introduction

In recent years, the public concern about safety has increased because of the increase in the number of terrorist attacks all over the world. Constructing civil and commercial buildings capable of sustaining impact and blast loads with acceptable damage has gained a lot of attention. Blast attacks are one of the main sources of impact load because of the fragments and debris associated with an

WIT Transactions on The Built Environment, Vol 126,
www.witpress.com, ISSN 1743-3509 (on-line)
doi:10.2495/SU120251

explosion of a charge weight. The velocity of the impact governs the response of the structure. Low-velocity impacts may cause quasi-static response, while hyper-velocity impacts can cause the properties of the material to change [1]. In other words, the dynamic response of structural components subjected to short-duration impacts is different from other type of loads. Impact Load is a relatively large dynamic load applied to the structure or part of the structure in a comparatively short period of time [2].

In this paper, the focus will be on numerical modelling for the structural component of the building façade in the form of reinforced concrete panel under impact. The work studies how these panels can exhibit local damage that reduces the forces transmitted to the structural component and thus avoid collapse and sudden failure. This work is a continuation of previous work done by the authors [3]. A simplified time history analysis simulating impact load was. In this paper, the developed model is further examined to identify the parameters such as the mesh size, time step, and material properties that affect the model accuracy. The developed and calibrated model is further used for detailed parametric study.

2 Experimental setup

The numerical models are validated against the results of previous experimental work done at the American University in Cairo [4].

2.1 Apparatus

The impact apparatus used in the experimental setup [4] is a pendulum type. The apparatus, as shown in Fig. 1, consists of: winch and winch support; specimen's supporting frame; and Pendulum mass, pendulum supporting system. The winch is used to lift the mass to the required heights. The apparatus was designed to hold different specimen elements such as RC panels, columns, slabs in addition to beams.

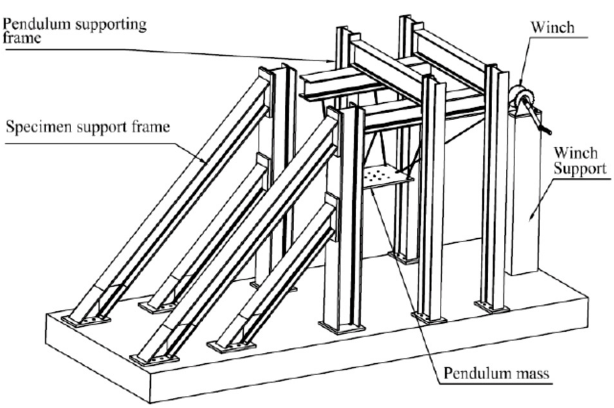

Figure 1: Impact test apparatus [4].

2.2 Specimens

The reinforced concrete specimen is of cross-section of 300×150 mm and the length 1100 mm. The beam was reinforced by 2 bar size 10 mm at the top and 2 bar size 10 mm at the bottom. The beam stirrups were bar size 8 mm every 150 mm. The concrete compressive strength was 40 MPa after 28 days.

3 Numerical model

LS-DYNA is selected to be the numerical code in this research. Three numerical packages are used in this research, namely ANSYS V12.0, LS-DYNA Program Manager, and LS-PrePost 3.1 [5, 6].

3.1 Applied loads

Impact loads can be modelled numerically either by time-history function (simplified impact analysis) or real-time impact (simulated pendulum analysis) with reasonable accuracy. Although it is much easier and less expensive to model impact load as time-history function rather than real-time impact analysis, the real-time impact analysis better simulates the experimental setup.

3.2 Material models

For modelling concrete, Material-#072R3 (MAT-CONCRETE DAMAGE-REL3) was used. It is a three-invariant model, uses three shear failure surfaces. Material-#072R3 includes damage, strain-rate effects, and origins initially based on the Pseudo-Tensor Model (Material-#016). Model parameter generation capability is the major user improvement provided by Release III [6].

For modelling steel plates and reinforcement bars, Material #024, MAT_PIECEWISE_LINEAR_PLASTICITY was used. It is an elasto-plastic material model with an arbitrary stress versus strain curve; moreover, arbitrary strain rate dependency can be defined. Also, failure based on a plastic strain or a minimum time step size can be defined.

3.3 Model development and calibration

Actual time history of impact forces were measured using fatigue rated dynamic load cell attached to the pendulum mass and recorded using high speed data acquisition system [4]. Those values are used as input force for the numerical model. Figure 2 shows the input forces used as a preliminary model used for model development. The front steel plate attached to the RC beam was loaded by time-history function for a mass of 142.87kg falling from a height of 100 mm while the beam response was presented.

3.4 Mesh size

Figure 3 shows the solid elements used for concrete beam and impact mass meshed every 25 mm. Steel plates were meshed by 25×25×10 mm. Steel bars

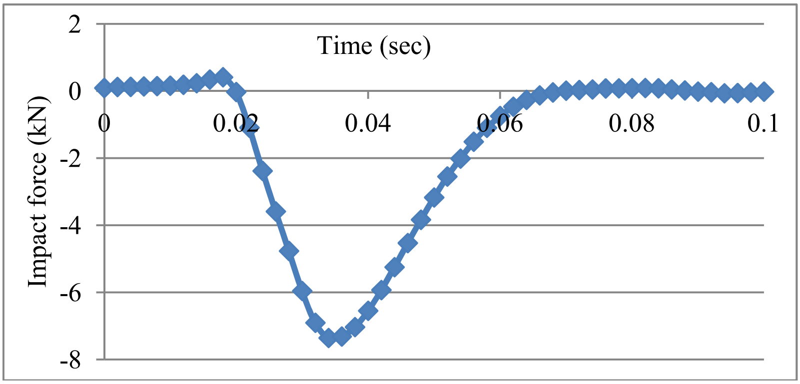

Figure 2: Input force for a Mass of 142.87 kg failing from a height of 100 mm.

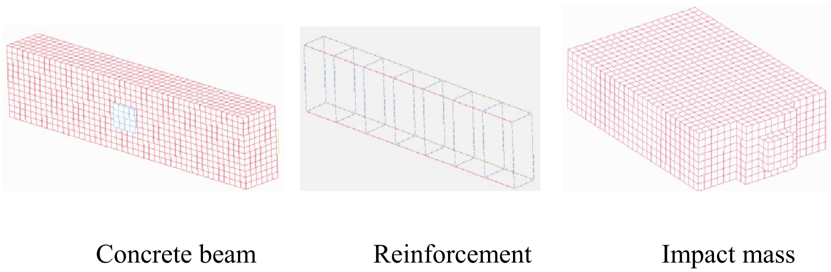

Figure 3: Finite element meshing for different components.

were meshed every 25 mm. All elements were tied together at the intersecting joints through shared nodes. Different mesh sizes are not preferred to avoid numerical errors.

The mesh element size has been selected after several trials considering the recommendations reported in previous works [7, 8]. Another governing factor was the geometry of the beam used in the experimental study. For example, Tahmasebinia [7] recommends the use of 19 mm element size but since concrete cover 25 mm and stirrups spaced every 150 mm, mesh size of 25 mm is used [9].

4 Analysis of results and discussions

The time-history function for a mass of 142.87kg falling from a height of 100 mm was modelled as a simplified pendulum analysis method. This preliminary model was used to check the numerical model accuracy [3]. Moreover, the preliminary model was used to identify the numerical model parameters such as the mesh size, time step, and material properties because of its shorter running time of 30 min. compared to the full dynamic simulation of falling mass that takes around 5 hours.

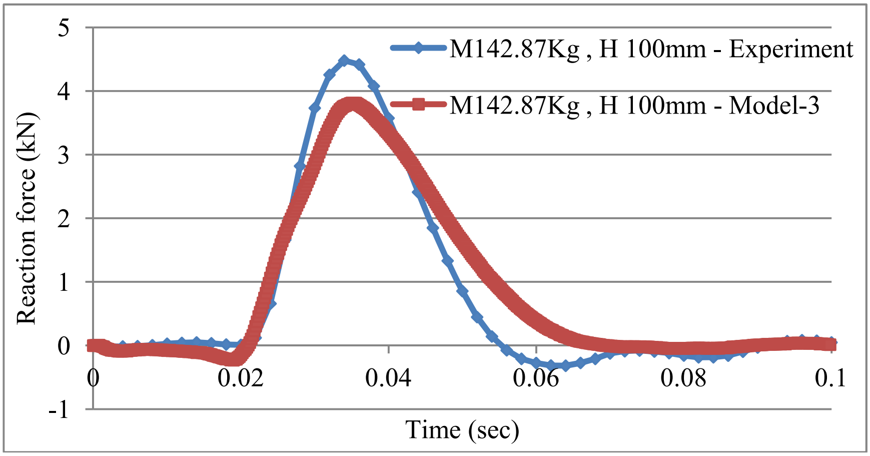

Figure 4: Reaction values of the simplified time history analysis.

The preliminary numerical results have shown good agreement with the experiment results. Quantitative analysis will be further examined under real simulated pendulum analysis instead of the impact input load function that does not account for the contact between the impact mass and the specimen.

As shown in Figure 5, the numerical model of simulating the full pendulum motion has shown good correlation with the experimental results in terms of the peak value of load and reaction except at small drop height. The maximum observed differences are 6.52% in the reaction values and 17.33% in the impact force values for higher drop height. The impact force of the numerical model was

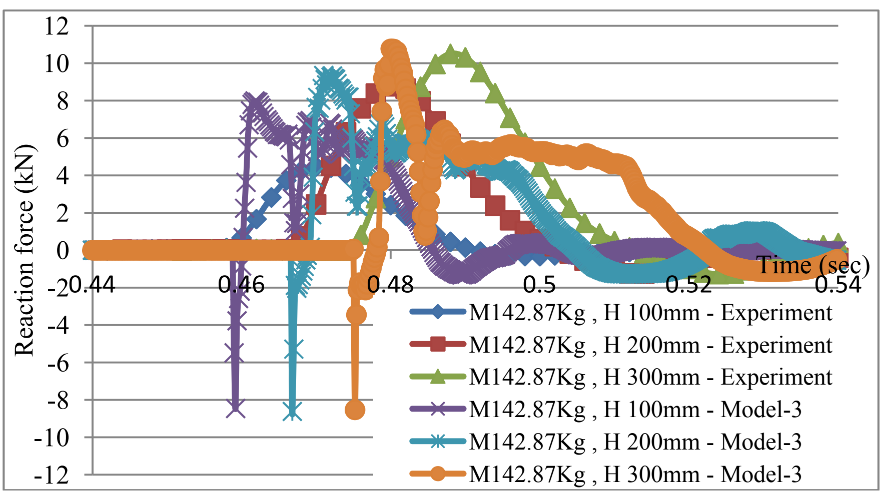

Figure 5: Simulated pendulum analysis vs. experimental reaction results for different drop height.

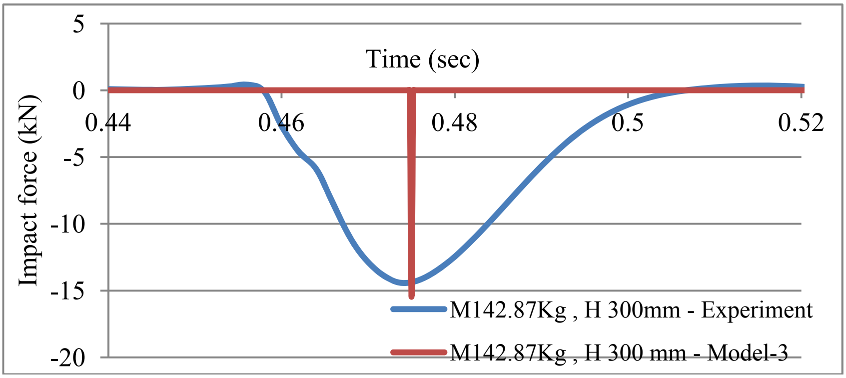

Figure 6: Simulated pendulum analysis – Model-3 vs. Exp. results (force).

greater than the experimental impact force by 17.33%. However, the duration of impact in the numerical model was 0.8 ms while it was 50 ms in the experiment testing as shown in Figures 4–17.

Although the numerical models have shown close values to the exp. results, this could be miss-leading because the numerical impact load profile was near pulse compared to the impulse shape of the experimental results. The dynamic modification factor of the numerical model (pulse load) is expected to be less than that of the experimental model (impulse load). This should lead to smaller values for the numerical model. Other factors not taken into considerations that could have affected the values of the numerical model are:

- Losses in the experimental setup such as the friction of the wench
- Movement of the supporting frame
- Deformations of the steel plates
- Local damage of the RC beam (micro-cracks) due to repeated impact on the same specimen.

5 Model refinements

Although the numerical model was initially meshed with a comparatively small element size (25×25×25) according to Tahmasebinia [7], two other mesh patterns are used to study the effect on the behaviour of the RC beam. The two types of meshing elements used are: Mesh-A: 13×13×13 mm and Mesh-B: 25×25×5 mm. Meshes “A” and “B” showed reduction in the reaction values 11% and 19% respectively. The results in better correlation between the numerical analysis based on simulated pendulum and the experimental results.

6 Parametric study

In this parametric analysis the effect of different parameters on the beam response, in terms of reaction forces and mid-span deflection, is investigated. The control model is listed in Table 1.

Table 1: Control model used in the parametric analysis.

Impact mass	142.87 kg
Impact height	100 mm
Tension reinforcement	10 mm
Compression reinforcement	10 mm
Steel plates modulus of elasticity (E)	200,000 MPa

6.1 Effect of the tension-side longitudinal reinforcement

The influence of the increase in diameter of the tension-side longitudinal reinforcement on the response of the beam is examined using bar diameter of 12, 18, and 22 mm in addition to the control model (diameter 10 mm). The effect of the tension bar diameters on the reactions and mid-span deflection is shown in Figs. 7–9. With the increase in diameter of the tension reinforcement, there is a significant decrease (60%) in the mid-span deflection as shown in Figure 7. For example, the deflection dropped to 4.25 mm using bar size of 22 mm compared to 10.67mm using bar size of 10mm. With the increase in the diameter of the tension reinforcement, there is a significant increase (102%) in reaction force as shown in Figure 8. For example, the reaction significantly increase to 16.19 kN using bar size of 22 mm compared to 7.98 kN using bar size of 10mm. The beam was getting stiffer to attract more loads. Figure 9 shows a little change in the reaction-time history profile with the increase in the bar size of tension reinforcement. With the increase of the bar diameter, the area under the curve is getting less.

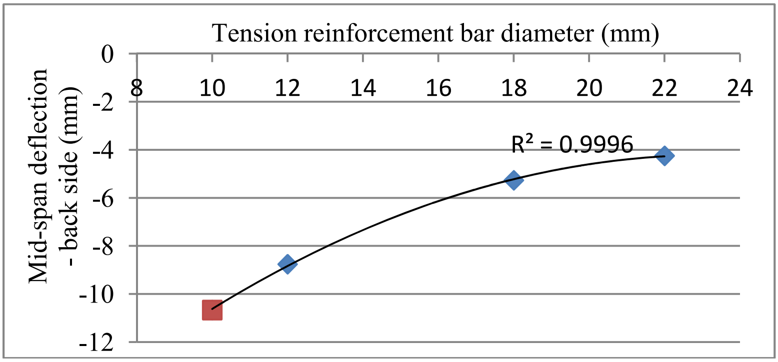

Figure 7: Effect tension reinforcement bar diameter on mid-span deflection.

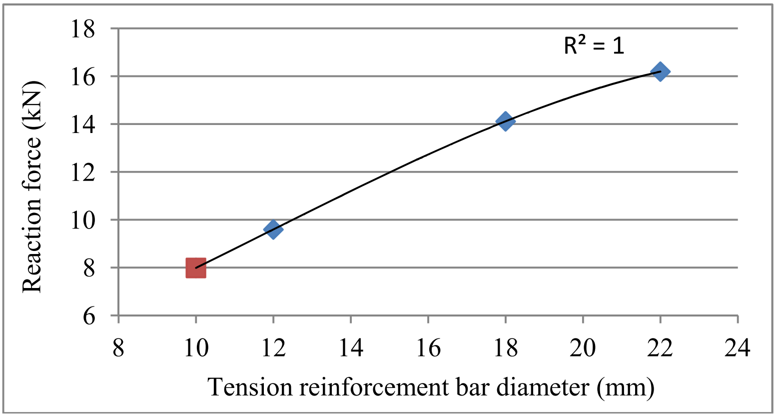

Figure 8: Effect of tension reinforcement bar diameter on reaction force.

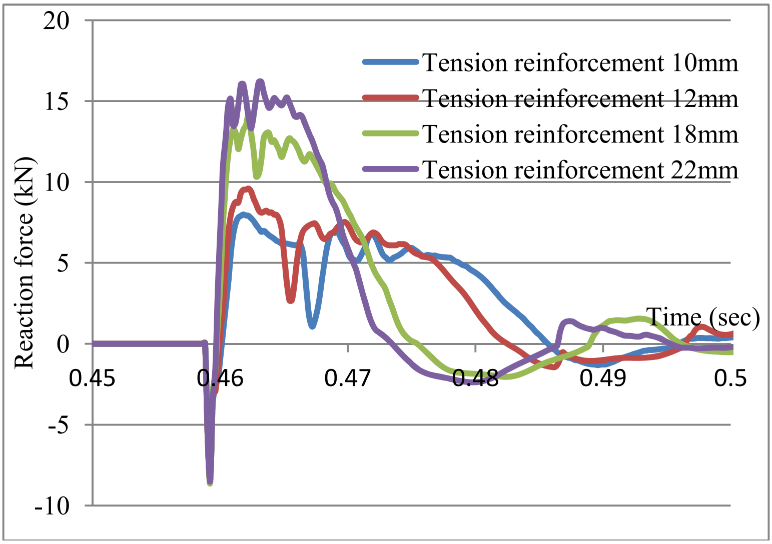

Figure 9: Reaction time history for different tension reinforcement bar diameters.

6.2 Effect of the compression-side longitudinal reinforcement

The influence of the increase in the diameter of the compression-side longitudinal reinforcement on the response of the beam is studied using diameters of 10, 12, 18, and 22 mm. With the increase in diameter of the compression reinforcement, there is a decrease (10%) in the mid-span deflection as shown in Figure 10. For example, the deflection dropped to 9.67mm using bar size of 22 mm compared to 10.67 mm using bar size of 10 mm. With the

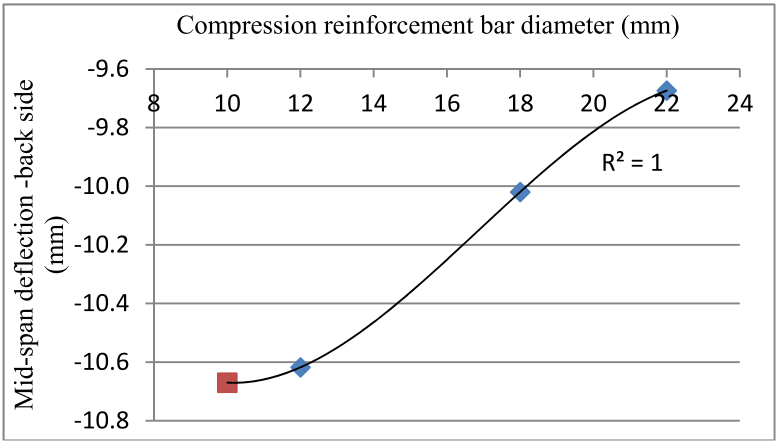

Figure 10: Effect of compression bar diameter on mid-span deflection.

increase in diameter of the compression reinforcement, there is no significant change in reaction forces.

6.3 Effect of the front plate elasticity

In order to study the effect of the elasticity of front plate; different materials can be used such as rubber and Teflon. However for simplicity, the modulus of elasticity (E) has been selected as an index to represent the elasticity of the front plate while keeping the same steel material. The influence of the front plate elasticity on the response of the beam is studied using E values of 20, 200, 2000, 20000 MPa, in addition to the control model (E = 200000 MPa). As shown in Fig. 11, as the modulus of elasticity of the front plate decreases, the impact duration increases at the same time the impact peak load decreases. For example,

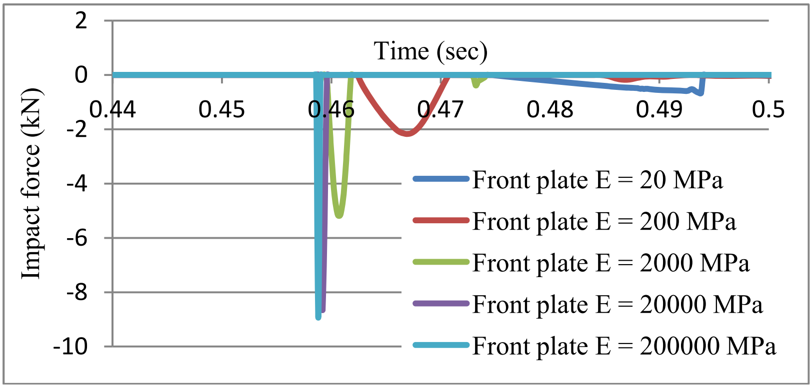

Figure 11: Effect of front plate modulus of elasticity on the load time history.

at E = 200 MPa the impact force was 2.22 kN and duration 8 ms compared to the impact force of 8.89 kN and duration 0.8 ms for E = 200000 MPa.

7 Conclusions

The numerical code, LS-DYNA, is successfully used to analyze the response of concrete panels under impact load through simulation of pendulum movement dropped from elevated height up to and including the contact impact and the nonlinear dynamic response of the RC panel to such impact load as well as cracking and damage simulation. The impact phenomena can also be modeled numerically using simplified analysis by having the short duration impact load history as input to the structural element. This allows also for analysis not only under impact load but also under the effect of blast load using generated blast load function. The numerical models were validated against the results of previous experimental work done at the American University in Cairo. Away from small impact force values at relatively small drop height, comparisons with experimental values showed that the Material Model used in the analysis provides the closest results to the experimental results with maximum differences of 6.52% in the reaction force and 17.33% in the impact force.

Nonlinear model simulating the pendulum movement up to impact showed that the impact load was near pulse values as the load duration of the numerical model was 0.8 ms which is in agreement with previous studies. The simulated pendulum analysis was able to simulate the reaction force with reasonable accuracy and thus can be used to design the façade panels and the supporting frame. In the current study the impact was applied at the mid-span point which is more critical for the panel. However, for the design of the supporting frame different impact location can be easily considered with the same approach.

Parametric analysis was carried out using the developed numerical model to study the effect of different design parameter on the response to impact. Mid-span deflection and stresses increase with the increase of the impact mass and height. Mid-span deflection decreases with the increase in the elasticity (flexibility) of the front and back steel plates. Accordingly, having an elastic layer in front of the building facade will reduce the response. Having a protective plate at the point of impact reduces the mid-span stress by allowing for better distribution of impact stress over a wider portion of the beam. This effect becomes more pronounced as the thickness of the plate increases.

References

[1] Jones, N., *Structural Impact*. Cambridge University Press, Cambridge, 1989.

[2] Schenker, A., Anteby, I., Nizri, E., Ostriach, B., Kivity, Y., Sadot, O., Haham, O., Michaelis, R., Gal, E., and Ben-Dor, G., Foam-Protected Reinforced Concrete Structures under Impact: Experimental and Numerical Studies, *J. Struct. Eng.*, ASCE, **131(8)**, 1233-1242, 2005.

[3] Abdel-Mooty M. and Shaaban S., Numerical Modeling and Calibration of Impact Testing of Reinforced Concrete Panels. Proceedings of the Tenth

International Conference on Computational Structures Technology, ed. B. Topping, J. Adam, F. Pallarés, R. Bru, M. Romero, Civil-Comp Press, Stirlingshire, UK, Paper 103, 2010. doi:10.4203/ccp.93.103, 2010.
[4] Cherif, H. Experimental Investigation Of Building Components Response To Impact And Short Duration Loads, M.Sc. Thesis, the American University in Cairo, 2009.
[5] ANSYS, (2009), Version 12.0, Help manual, ANSYS, Inc.
[6] Livermore Software Technology Corporation, LSTC, *LS-DYNA Keyword User's Manual: Volume I & II*, Version 971/ Release 4, Livermore, CA, 2009.
[7] Tahmasebinia, F., Finite Element Simulation of Reinforced Concrete Structures under Impact Accident, *Structural Survey*, **26(5)**, pp. 445-454, 2008.
[8] Sangi, A. and May, I., High-Mass, Low-Velocity Impacts on Reinforced Concrete Slabs, *7th European LS-DYNA Conference*, 2009.
[9] Shaaban, S., Nonlinear Numerical Modeling Of Reinforced Concrete Panels Subjected To Impact Loads, M.Sc. Thesis, the American University in Cairo, 2011.