Journal of Civil Engineering and Construction 2022;11(3):127-138
https://doi.org/10.32732/jcec.2022.11.3.127

Challenges in Modelling Reinforced Concrete Panels Subjected to Blast Load - A Critical Review

Amina Mohammed[1*], Husham Almansour[2]

1. Department of Civil Engineering, University of Ottawa, CBY A106 161 Louis Pasteur,Ottawa, ON, K1N 6N5, Canada
2. National Research Council Canada, 1200 Montreal Road, Ottawa, Ontario K1A 0R6, Canada
E-mail: amohamme@Uottawa.ca (Corresponding author)

Received: 5 May 2022; Accepted: 9 July 2022; Available online: 5 August 2022

Abstract: Reinforced concrete panels are widely used in modern facilities, and evaluating their blast loading capacity is vital for security-critical assets. Due to the high impulsive nature of blast loads, the response of reinforced concrete panels is characteristically different from that under static or low dynamic load conditions. The failure of individual components often initiates the blast load's destructive effects on the entire structure. The material breach can be caused by stress wave localized effects before the general structural response becomes significant. Numerical methods are one of the key methods for studying the behaviour of reinforced concrete panels under blast load. This paper aims to review the current state of practice in modelling reinforced concrete panels and predicting their blast capacity and failure mechanisms under blast load. The work addresses the research gaps associated with using advanced finite element modelling as compared to test results.
Keywords: Reinforced concrete panels; Blast load; Finite element modelling.

---

1. Introduction

The exterior envelope of a building is the most vulnerable part when subjected to extreme blast loads. Reinforced concrete (RC) panels are widely used in modern facilities, and evaluating their blast loading capacity is vital for security-critical assets. Due to the high impulsive nature of blast loads, the response of RC panels is characteristically different from that under static or low-frequency dynamic load conditions. This problem is complex as the instantaneous material's behaviour is difficult to control, and the mechanical behaviours vary under different blast load conditions [1]. It is generally understood that the blast load's destructive effects on structures are often initiated by the failure of individual components, where the material breach can be caused by stress wave localized effects before the general structural response becomes significant.

The methods used for studying the behaviour of RC elements/structures under blast load include analytical, experimental, numerical methods, and a combination of the three techniques [2]. The analytical methods are applied when simplified and approximate results are acceptable, where appropriately assumed conditions are made to solve the problem using a theoretical model after idealizing the shock wave propagation or impact load. The experimental methods are usually conducted in small-scale or full-scale prototype testing by selecting the proper effect parameters for the blast pressure wave. Then, the results are analyzed using the statistical regression method in order to obtain the appropriate empirical formula or monograph representing the structural dynamic response of the tested structural element or system. By using numerical analysis such as the finite element method or the finite difference method, the fundamental laws of mechanics such as mass, energy, and momentum are introduced in the dynamic response of the material and the failure criterion of structures [3]. Numerical models can explore the feasibility of RC structural concepts and their structural performance before field blast testing or advanced experimental investigation of more complex structures affected by blast loads is conducted. However, numerical models (mainly using finite element modelling (FEM) and/or computational fluid dynamics (CFD) packages such as LS-DYNA, AUTODYN, ABAQUS, etc.) are often complex and sensitive to some tunning parameters. The background assumptions and sequence of modelling steps, including discretization, complex operations, and refinements used to reach a solution, are often unknown to most users. Using commercial FEM/CFD software raises the challenge of ensuring the convergence of the models, the accuracy of their results, and the ability to generalize the application of the modelling approach to similar cases with different scales, geometry, or material properties. The numerical suites (loading functions, material models, and element types) available for solving various engineering statics and dynamics problems could not yield relevant results under all conditions [4]. Hence, there is a need to (i) comprehensively understand the theory behind different options offered by the finite element software packages to make the proper selections; (ii) satisfy the FEM requirements such as numerical stability and verification; (iii) validate the results obtained from the FEM with available experimental or at least with reliable analytical results since the complexity associated with modelling of composite materials such as reinforced concrete makes it is difficult to entirely rely on the results of a numerical analysis without validation [4].

In general, using finite element analysis presents a set of challenges such as (i) selection of a suitable problem-specific mesh, (ii) ability to examine the stability of the solution procedure, and (iii) possible sources of errors based on modelling assumptions, mesh refinement/number of degrees of freedoms, and the time step size, etc. Selection of the appropriate element type for numerical analysis is essential for an effective finite element model. The selection is based on the desired level of accuracy and the time and cost associated with each numerical method [5]. The compatibility of the selected element with the constitutive material model used in the numerical analysis procedure (numerical simulation) is also critical. On the other hand, a numerical simulation's primary goals are to match experimental results in deformations, predict crack/damage patterns, and simulate the general structural behaviour, considering that the simulation should satisfy finite element requirements such as convergence numerical stability. More importantly, a fundamental goal of an effective model is its ability to generate accurate results when the loads and boundary conditions are changed and when the geometrical and material parameters are generalized.

This paper aims to review the current practice in modelling RC panels and predict their failure mechanisms under blast load. It seeks to identify and address issues associated with using FEM to predict the behaviour of RC panels subjected to blast load. In addition, the study aims to identify the most comprehensive modelling procedure and fine-tuning processes to develop an effective model that can simulate RC panels with high accuracy for all loading cases and boundary conditions. The paper also presents the requirements for a numerically efficient and accurate FEM capable of predicting the behaviour and failure mechanisms of RC panels subjected to blast load.

2. Steps required for an effective finite element modelling

Commercial software packages have different material models for the concrete, steel, constraint, and blast load. They also have different options to set up the erosion, control the hourglass, etc. These wide varieties of modelling options can be possible sources of errors based on modelling assumptions. Therefore, attention needs to be carefully taken to satisfy the requirements of effective numerical simulations in terms of stability, verifications, and validations. Figure 1 shows the proposed comprehensive procedure steps required to develop an effective FEM of RC panels subjected to blast loading that satisfy the requirements of adoptable numerical simulations. The procedure is based on the fundamental concepts of FEM offered in many original references, for instance, Zienkiewicz [6]; Gallagher [7]; Burnett [8]; and Bathe [9].

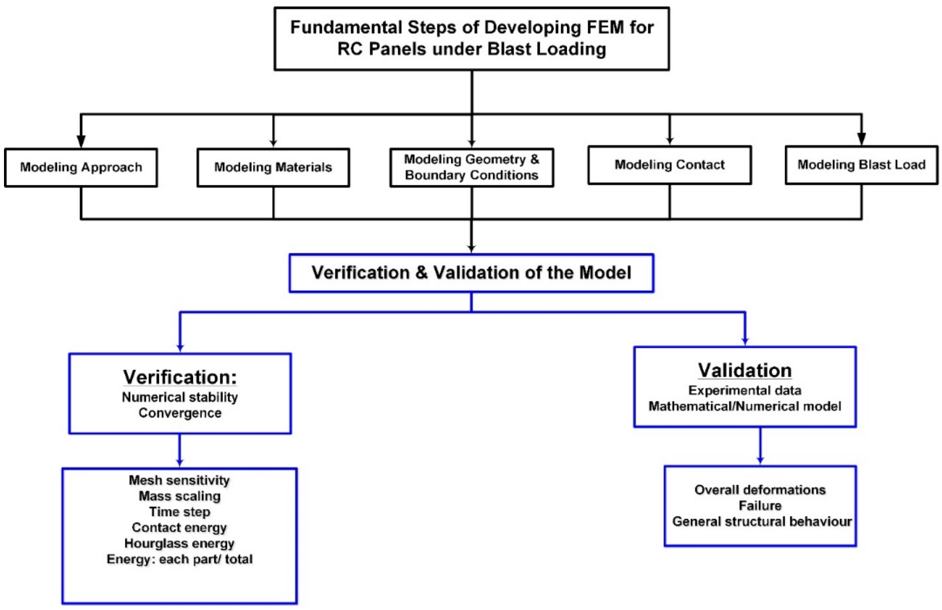

Figure 1. Effective FEM procedure for RC Panels Subjected to Blast Load

2.1 Modelling approach

Several commercial software packages, such as ABAQUS, AUTODYN, EUROPLEXUS, DIANA, or LS-DYNA offer different ways to address numerical simulations of blast-loaded structures. This section describes different modelling approaches that have been considered in previous studies and various options available in some finite element packages (mainly in LS-DYNA) that had been selected or chosen in their models.

Using ANSYS AUTODYN software, Abdel-Mooty et al. [10] conducted nonlinear dynamic numerical modelling of concrete panels subjected to blast loads. They validated the accuracy of the model against experimental results of blast loading response of reinforced concrete panels conducted earlier by Razaqpur et al. [11]. Using the ANSYS AUTODYN solver (ANSYS V.14.5 – Explicit dynamics system), the model is composed of three parts: (i) the concrete body has 1000×1000×70 mm dimensions, (ii) the longitudinal reinforcing steel, and (iii) the transverse reinforcing steel. The reinforcing steel rebars in both directions are of a cross-sectional area of 25.8 mm^2. Solid elements are used in modelling both the concrete body and the reinforcing steel rebars. The characteristics of the solid elements govern the meshing method and its refinement. The mesh physics preference is set to explicit with coarse relevance centre and triangle surface mesher program-controlled, which leads to tetrahedrons elements. To imitate the same boundary condition, the model is assigned to have fixed supports all over the panel edges. The assigned load type is chosen to be a uniform pressure equal to 5.059 MPa with a time duration equal to 7.7×10^{-4} sec. The numerical model is verified and used to (i) investigate the strain field on the surface of the concrete panel, the kinetic energy, the internal energy, and the total work, and (ii) to carry out a parametric study including the concrete compressive strength, concrete panel thickness, boundary conditions, reinforcement steel quantity, and steel reinforcement arrangement [10].

Galuta and Regig [12] conducted a numerical model using AUTODYNE code v. 17 to study the erosion effect on the numerical simulation of RC panels subjected to the impact of the high-speed projectile. The RC panel has 600 × 600 x 100 mm dimensions and concrete compressive strength of 35 MPa. The panel is reinforced with 10 mm diameter hot rolled deformed steel bars in two perpendicular directions spaced at 200 mm c/c. A 60 mm ogive-nosed bullet (.50 caliber barrel) is used to hit the concrete panel at the center with an initial velocity of 640 m/sec. The boundary conditions for the RC panels are assumed to be fixed. In Galuta and Regig [12] simulation, the variations of erosion strain values and mesh size of the concrete panel on the response of the bullet residual velocity and damage depths at the front and rear faces of the concrete panel are investigated. The bullet residual velocity and damage depth of the concrete panel are obtained and compared with experimental results. Major findings of the study are that the bullet residual velocity and damage pattern of the concrete panels are strongly dependent on erosion value and mesh size. The residual velocity of the bullet is remarkably reduced as the erosion strain value is increased. However, using large erosion values in the numerical model could result in an excessive mesh distortion.

Tai et al. [3] conducted a simulation using LS-DYNA on an RC slab that has 3.6 m length, 3.0 m width, and 0.15 m thickness; the slab is reinforced with 18 steel bars in the short direction and 15 in the long direction. The explosives were placed 2.5 m above the center of the slab. Due to the RC slab and applied load symmetry, half of the model is taken for the simulation to simplify and reduce the computation efforts. During the simulation, the RC slab has fixed boundaries as the slab is divided into small slabs stiffened by beams; the edges of the slab can provide rotational restraint due to the torsion rigidity of the beams. The effects of artificial reflections at open boundaries can severely reduce the accuracy of the computations; hence, Tai et al. [3] assumed a non-reflecting boundary for the air medium. Thus the pressure is assumed to flow out of this boundary and not cause any reflection. The concrete is modelled using hexahedron continuum elements with what is called Johnson–Holmsquist Concrete for the concrete material model, while beam elements were used for the steel reinforcement. Tai et al. [3] employed the steel elastoplastic constitutive law. The isotropic hardening rule was used as the hardened model, and the extreme plastic strain defined the steel material failure. Once the element's strain approaches this plastic strain in the computation procedure, the element is regarded as failed and eroded. In Tai et al. [3] work, the discussion mainly focuses on the meshing of blast pressure, the size of explosives, and distance of explosives from the slab. The amount of reinforcement is also considered. However, the effects of the mesh size of the slab and the reinforcement are not considered. The authors concluded that the mesh size is very sensitive to shock wave propagation in simulating the free-field blast pressure wave. The study recommended that the finite element mesh division should be as fine as possible to ensure approaching close results to the actual loading and performance state. It is important here to mention that the authors did not consider the numerical cost of the extreme mesh refinement.

Alañón et al. [13] constructed and modelled RC slabs with a cross-sectional dimension of 4400 mm × 1460 mm and a thickness of 150 mm. The slab is reinforced with a 12-mm diameter rebar meshing spaced at a distance of 150 mm from one another in the major bending lane. The charge weight is 15 kg TNT equivalent explosive charge at a specific standoff distance of 1 m, Alañón et al. [13]. The concrete slab is modelled by a Lagrange mesh, in which the coordinates move with the material. For the slab supporting area, upper and back supports are created, and all the displacements of the supports are assigned as fixed [13]. Due to the symmetry of the structure, only the longitudinal half of the slab is considered, discarding the transverse plane of symmetry to fully match the most affected section of the slab. In Alañón et al. [13] simulation, 3D solid elements are assigned for the concrete; these elements are eight-node hexahedral with three degrees of freedom (DOF) per node (three displacements). In order to reduce the computation time, linear shape functions and reduced integration into a single point per element (constant stress solid element) are taken into account, including Hourglass control to prevent deformation modes with zero energy [13]. For the reinforcement rebars, bar elements are used with the Hughes-Liu LS-DYNA formulation based on degenerated eight-node solid elements. The material model for the concrete is implemented as the Continuous Surface Cap Model (MAT-CSCM-CONCRETE) (MAT159), and for the steel is PIECEWISE-LINEAR-PLASTICITY material. Alañón et al. [12] explored the reliability of finite element simulations of an RC slab under a certain level of explosion; the focus was on considering the mesh size sensitivity on the RC slab behaviour in terms of damage pattern (which was compared to experimental results) and internal energy history. Fifteen finite-element simulations were evaluated with coarse to fine range mesh sizes of the concrete, and the effects of mesh resolution on the numerical simulations were analyzed. However, the effects of concrete mesh size are only considered ignoring the effects of steel bar mesh size.

Castedo et al. [14] conducted a series of blast tests on eight RC slabs to calibrate and validate their numerical model results. They tested an RC slab of 4.40m X 1.46m X 0.15 m reinforced by 12 mm diameter steel rebars at 150 mm C/C in the longitudinal and transverse directions at the top (facing the load) and at the bottom faces, where the concrete cover was 30 mm. The slab was covered by a 1.5 m X 1.46 m, 10 mm thick steel sheet on its top side at mid-span. Due to specimen symmetry and the simulation time cost, one-half along the central axis of each slab is considered. The boundary condition for the concrete and steel material is defined within the symmetry plane as no displacement going through it [14]. Also, all displacements are fixed in the slab supporting area at the bottom face to reproduce the test behaviour. In the simulation, concrete was discretized with 3D Lagrangian solid elements, while the steel reinforcement was modelled using beam elements, assuming a perfect bond [14]. The rebar was incorporated into the concrete mesh using the constrained method. Two concrete material models were used: Johnson Holmquist Concrete (or JHC) and CSCM concrete; these models need minimal input parameters and account for different damage behaviours in tension and compression [14]. The reinforcement steel bars and the steel sheet are included in the model by the piecewise linear plasticity material model treating the stress-strain by a bilinear curve including strain rate effects. In Castedo et al. [14] work, the major validation was done in terms of damage pattern. They claimed that the different modelling setting effects on the results are rarely reported, which makes it necessary to have more numerical studies to understand the complexity of dynamic response of reinforced concrete structures. However, they did not mention how they have dealt with convergence/sensitivity and stability issues, as they are major keys to any numerical model's accuracy.

Maazoun et al. [15] modelled the blast response of RC hollow-core slabs that have a thickness of 215 mm, a width of 600 mm, and a length of 6000 mm, with the explosive charge of 1.5 kg of C4 suspended under the slab at a standoff distance of 0.5 m. Due to the symmetry of the panel geometry and blast loading (taking into account the time cost of simulations), a quarter slab is adopted and considered for modelling. The numerical model is analyzed for 0.5 seconds, which is sufficient for the blast wave to propagate throughout the slab. For the quarter model, the translation of the nodes in the symmetry planes is constrained [15]. In the simulation, the concrete is modelled using constant stress solid elements, and for the steel, the reinforcement consists of Hughes-Liu beam elements. Two material models of concrete are chosen; the Winfrith concrete model and the Concrete Damage Release 3; and the steel rebars are modelled using PIECEWISE LINEAR PLASTICIT to represent steel reinforcement behaviour, with plastic deformation, strain rate effects, and failure. A dynamic increase factor (DIF) was applied for both yield and ultimate strength of the steel reinforcement, and the reinforcing beam elements were coupled to the concrete elements through constrained Lagrange in the solid formulation neglecting bond slip. In order to contain perfect interaction and bond, duplicated nodes between the compression layer and the hollow core slab were merged. Maazoun et al. [15] described how to develop a numerical model to predict the maximum deflection and crack distribution, but did not mention how they have dealt with convergence/sensitivity issues and model stabilities. It is stated that the accuracy of FEM is related to correctly predicting the maximum defection and crack distribution compared with experimental results. However, other parameters such as hourglass energy, erosion value, and material models are only considered to evaluate their effects on the structural response of the considered structure.

In Wenjiao et al. [16], a one-way reinforced concrete slab with a size of 1300×1000×120 mm was modelled to study the dynamic response of RC structure under blast loading. The RC slab is placed on a steel frame, and the two short sides of the RC slab were clamped to prevent uplifting during the test. Wenjiao et al. [16] applied a blast load developed by a detonation of 2.09 kg trinitrotoluene (TNT) explosive charges at a 600 mm standoff above the specimen. In order to reduce the risk of stress concentration at the specimen boundary, four wooden blocks were inserted between the specimen and the steel supporting frame. The concrete slab was modelled using a solid element, an 8-node constant stress hexahedron brick element in the simulation. Reinforcement bars were modelled using a beam element. No-slip between the steel reinforcement and concrete is assumed, Wenjiao et al. [16]. However, in real cases, a bond-slip between steel and concrete exists; it was found that the perfect bond assumption had a small influence on the result of RC structures under blast loading. The Lagrangian formulation in which the coordinates move with the material was applied in the analysis. An RC slab was supported on two rigid plates made of solid elements to simulate the experimental conditions. The element size of the concrete and steel is defined as 5×5 mm after verifying the convergence of the grid. The axial symmetry with respect to the central line of the RC slab is taken into account. Thus only 1/4 of the slab is modelled to reduce the computation time. Concrete Damage Rel3 is used to simulate the concrete material as it can reproduce key concrete behaviours vital to blast analyses. It is also easy to calibrate using laboratory data. The material model of PLASTIC KINEMATIC is used for the steel reinforcement because it is fitted for simulating isotropic and kinematic hardening plasticity with consideration of the rate effects. A dynamic increase factor (DIF), which is the ratio of the dynamic-to-static material strengths versus strain rate, is considered to realize the simulation of RC slab under blast loading [16].

A general contact algorithm modelled the interaction between support plates and RC slab, which uses a penalty contact method. This method searches for small nodal penetrations between surfaces where contact occurs. Automatic single-to-surface contact is used to simulate contact between the support plates and the RC slab. In order to simulate the concrete damage under blast/impulsive load, including shear failure, cratering, spalling, and crushing, the erosion algorithm is used. Wenjiao et al. [16] simulation aimed to get dynamic data of the cracked RC slabs under blast loading. Constraints of steel and concrete are not clearly stated in the paper. Dynamic increase factor is included. Bond slips between steel bar and concrete are assumed to be perfect, which is not the same as the actual state. Hence, requirements for more robust and accurate numerical algorithms are needed.

Mosalam and Mosallamb [17, 18] used the finite element platform DIANA 7.2 to develop computational models of RC slabs with and without CFRP composite strips subjected to static and/or dynamic loadings. The authors developed computational models based on an isoperimetric degenerated-solid approach using the finite element method incorporating eight-node quadrilateral shell elements. The used shell elements were numerically integrated into the plane of the shell using 2x2 Gauss integration and across its thickness using five integration points. The models are used to investigate different parameters, including load duration and the effect of CFRP retrofit on the damage accumulation. For the application of FRP composite strips, a layered version of the shell elements was used. In this case, the thickness of each element was subdivided into several layers where each layer had its own material properties and numerical integration scheme. The steel reinforcement mesh was modelled using an embedded grid with equivalent thickness. The reinforcement grid was numerically integrated using 2x2 Gauss integration [17,18]. These embedded grids add stiffness and strength to the concrete finite elements. The concrete material was modelled using total strain formation following the coaxial stress-strain concept, where stress-strain relationships are evaluated in the principal directions of the strain vector. The reinforcement material was modelled assuming elastoplastic behaviour based on Von-Mises yield criterion with a modulus of elasticity, yield strength, and 33% hardening utilizing the work hardening hypothesis [17,18]. It is claimed that comparing between the experimental and the analytical results indicated the validity of the computational models in capturing the experimentally determined results for both the control and the retrofitted slabs. The major findings of this work is comparing between the experimental, and the analytical results indicated the validity of the computational models in capturing the experimentally determined results for both the control and the retrofitted slab.

RC panels with openings have not been widely considered. Mays et al. [19,20] developed a method for the derivation of quasi-static elastic/plastic resistance functions for reinforced concrete wall panels with openings based upon finite element analysis and yield-line theory. They aimed at correlating a theoretically defined resistance-deflection function with model-scale quasi-static tests. In their work, blast loading trials were conducted on model reinforced concrete panels with openings to validate experimentally the yield line patterns predicted by a single degree of freedom analysis based upon a dynamically enhanced bi-linear resistance-deflection function. The trials showed that, in general, the location of the crack was similar to that observed in equivalent statically-loaded panels and reported in Mays et al. [20]. Damage levels in all blast-loaded panels were consistent with support rotations less than those predicted. Considerable numerical work needs to be carried out to demonstrate the ability of RC panels with openings to resist blast loadings.

2.2 Modelling blast loads

Blast loading can be simulated with a number of methods. The methods and their features are listed below (LS-DYNA 2017[21], and Tabatabaei et al. [22])

1) Lagrangian methods: consist of total Lagrangian and updated Lagrangian formulation; in this method, the mesh is deformable, and nodes and material points are at the same location.

2) Eulerian methods: widely used in fluid dynamics; in these methods, (i) the computational mesh is fixed, and the continuum moves through the mesh; (ii) in the Eulerian description, large distortions in the continuum motion can be handled with relative ease; and (iii) the mesh is fixed and does not deform, the material can move from one element to the next.

3) Arbitrary Lagrangian-Eulerian (ALE) methods: in these methods, (1) the mesh does not distort; (2) mesh contracting or expansion is possible; (3) translation or rotational mesh motion is allowed; and (4) the material can move from one element to the next.

4) Smooth Particle Hydrodynamic Methods (SPH): The methods work by dividing the domain into a set of discrete particles. The particles have a spatial distance, known as smoothing length, over which a kernel function smoothes the particle properties.

5) Element Free Galerkin methods (EFG): No mess is needed (meshless methods), and shape functions are constructed from sets of particles. Element Free Galerkin methods are based on moving least square approximations and use only a set of nodal points and a CAD-like description of the body to formulate discrete models. Normally they require a faster CPU than other methods. EFG methods cannot handle problems with severe deformations.

6) Discrete Element Methods (DEM): include a family of numerical methods for computing the motion of a large number of particles of micrometer-scale size and above. All particles are linked to their neighbouring particles through bonds. The properties of the bonds represent the complete mechanical behaviour of solid mechanics. The bonds are independent of the DES (Discrete Element Sphere) model. Bonds are calculated from the bulk modulus and shear modulus of materials.

2.3 Modelling materials

Shukla et al. [23] summarised a wide variety of concrete material models used for better performance of the reinforced concrete structure subjected to blast loading. Among these models, the most commonly used are: Concrete Damage-REL3, Winfrith Concrete, CSCM Concrete and Johnson Holmquist Concrete.

In the Concrete Damage-REL3 (MAT072R3) model, a three-invariant model that uses three shear failure surfaces and includes damage and strain rate effects allows the automatic generation of all the parameters by introducing the unconfined compressive strength and the density of the concrete. MAT072R3 is not able to model crack generation; therefore, the add-erosion material is used to model the failure. Winfrith Concrete (MAT084) uses a smeared crack model and smeared rebar model implemented in the eight-node single integration point continuum element. The maximum aggregate size and the fracture energy need to be defined if the strain rate is considered. MAT084 has a crack generation capability, and viewing the cracks is possible using a post-processing command. It is unsuitable for modelling confinement coming from reinforcement and FRP. Both MAT072 and MAT084 include strain rate effects and can predict the local and global response of concrete structures subject to impact and blast loadings, (LSTC 2017b[24]).

Johnson_Holmquist_Concrete (JHC model or MAT111) needs more inputs than the unconfined compressive strength. To identify each material parameter used in the model, triaxial compression and high strain rate dynamic tests must be performed on the concrete samples. The JHC model considers the material as linear elastic before a prescribed failure criterion is reached. Damage will accumulate upon further loading until total failure occurs, beyond which the material maintains a residual state. The post-failure strength surfaces are calculated by reducing the cohesion strength value of the initial failure surface. However, CSCM Concrete (MAT159) has a much simpler input, and it is suitable for low confinement conditions; it can simulate all effects and damage-based softening [24].

It is observed that the concrete damage model as the concrete panel under specified blast load could reasonably represent the behaviour of concrete at a high strain rate inadequate. Therefore, researchers recommended using the concrete damage model for most numerical simulations to represent concrete under the action of blast loads.

For simulating material properties of steel, it is noticed that many researchers have used plastic kinematic and piecewise linear plasticity material models. Various researchers carried out many parametric studies to investigate the effect of various reinforcement ratios, thickness slab, different charge weight, and standoff distance.

The plastic kinematic (MAT_003) model is suited to model isotropic and kinematic hardening plasticity with the option of including rate effects. Strain rate is accounted for using the Cowper and Symonds model, which scales the yield stress with the factor. Kinematic, isotropic, or a combination of kinematic and isotropic hardening may be specified by varying β' between 0 and 1 (LSTC 2017b[24]), as illustrated in Figure 2.

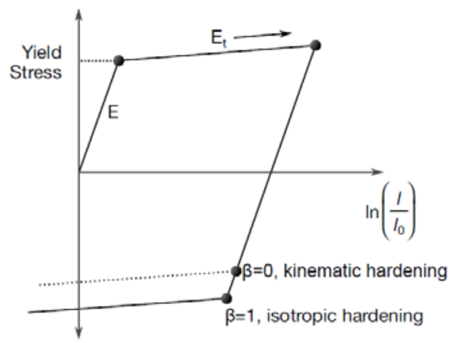

Figure 2. Elastic-plastic behaviour with kinematic and isotropic hardening (LSTC 2017b[24])

The Piecewise linear plasticity (MAT_024) model is an elastoplastic material with arbitrary stress versus strain curve and an optional arbitrary strain rate dependency. Failure can be defined as a function of plastic strain or a minimum time step size. The material description contains Young's Modulus, Poisson's ratio, yield stress, hardening modulus, ultimate plastic strain, and time step size for element deletion. A bilinear stress-strain curve may treat the stress-strain behaviour by defining the tangent modulus (ETAN). Rate effects may be accounted for by defining a table of curves. Strain rate effects follow the Cowper and Symonds model, which scales the yield stress with calculated factors based on the strain rate and strain rate parameters [24].

2.4 Modelling contact between materials

Figure 3 shows different methods of including rebar in a reinforced concrete element. The figure shows that reinforcement in concrete can be modelled as a layer of 'smeared' material; this model works best for small deformations where the reinforcement remains elastic or does not stress too far beyond the yield stress. On the other hand, when the reinforcement is to be included explicitly, the choices are: shared (merged) nodes or constraint methods. Shared nodes require the nodes of the reinforcement grid and concrete mesh to be identical. However, the shared node meshing becomes a challenge when three-dimensional reinforcement grids are required (multiple layers of reinforcement connected via additional reinforcement or stirrups through the thickness). In constraint methods, the reinforcement and concrete meshes are constructed independently. The two meshes are then superimposed in the appropriate relative geometric configuration. LS-DYNA internally constructs a system of constraints restricting the motion of the two meshes to be consistent. There are two options of constraint methods: (i) ALE COUPLING NODAL CONSTRAINT (ACNC), which has simplified input; and (ii) CONSTRAINED LAGRANGE IN SOLID: (CLIS), which is intended to provide a coupling mechanism for modelling Fluid-Structure Interaction (FSI).

CONSTRAINED LAGRANGE IN SOLID model has been used extensively to simulate bonding of reinforcing steel within the concrete, for instance(Tai et al. [3], Castedo et al. [4], Maazoun et al. [15] and Kiger et al. [25]). The main benefit of using this type of constraint is the flexibility of defining different meshes for the steel and concrete as nodes for these elements do not need to coincide. However, Schwer [26]) reported that using CONSTRAINED LAGRANGE IN SOLID model may cause errors in energy balance.

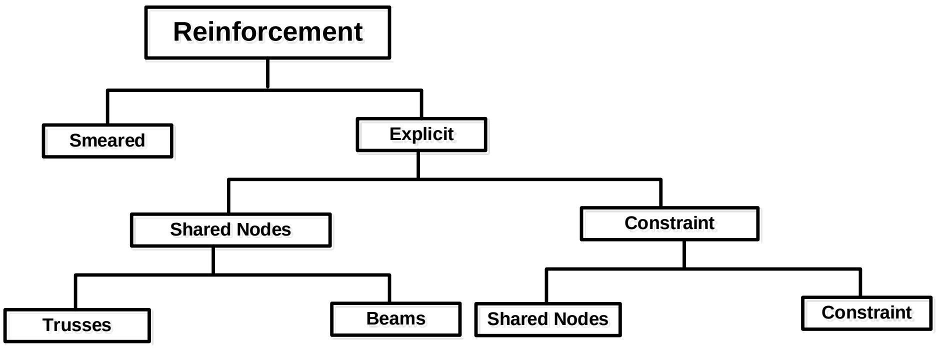

Figure 3. Schematic of various reinforcement modelling methods for rebar in reinforced concrete [26]

3. Verification and validation requirements of numerical models

Verification of FEM is the process of checking that a numerical model accurately represents a fundamental concept and specifications, while validation of FEM is the process of predicting real responses to a system compared to experimental data or analytical solutions [27].

In terms of verifications (see Figure 1), a general simulation must (i) ensure basic laws of mechanics such as conservation of energy, mass, and momentum, and (ii) check and confirm the basic physics represented by the model. Verification matrices of a numerical simulation have been set as follows (i) total energy must be conserved ±5% of damping/kinetic energy to internal in quasi-static problems; (ii) energy in each part must balance ±5; (iii) hourglass energy in each part must be set ±10; (iv) negative and positive contact energy must be set ±10; (v) time history sampling rate must ensure that the time histories yield correct data such as acceleration integrate to correct velocity; and (vi) mass scaling must not affect the solution and must be 3% [27].

With respect to validation (see Figure 1), numerical simulation results must match deformation, fracture/failure, and timing of experimental results. Two types of validation are considered: qualitative and quantitative validations. When the finite element simulation replicates the basic phenomenological behaviour of the actual full-scale test, it is called qualitative validation. And if the FE result values such as deformation, stress, and strain must be compared to those obtained from the test/experimental simulation, this is called quantitative validation [26]. In case of lacking experimental/field results, it is recommended to conduct the validation by comparing the simulation results to more than one finite element analysis program.

FEM can be used to explore the feasibility of structural concepts prior to explosive testing and to model more complex structures affected by blast loads. However, many challenges may be faced using finite element models, such as (i) possible sources of errors based on modelling assumptions and (ii) the ability to examine the stability of the solution. In finite element models, selecting a suitable problem-specific mesh, the appropriate element type (beam, solid, shell), and the appropriate material models are essential and are based on the desired level of accuracy and the time cost associated with each numerical method. The compatibility of the selected element type, mesh size, and constraint method with the constitutive material model used are very important for the validation and stability of FEM. Although there is a significant advancement of using 3D FEM, the effects of different modelling settings on the results are rarely reported; therefore, more numerical studies could be conducted to understand the complexity of dynamic response of RC structures.

3.1 Convergence study

Mesh convergence, or mesh sensitivity, is considered an important issue in verifying and validating a FEM where complex material behaviour, including damage and/or failure, is prevalent. Namely, the simulation has significant mesh dependency when material failure is invoked with the failure criteria.

In Tai et al. [3] , the discussion was mainly focused on the dynamic response of an RC slab under blast load where one-quarter of the simple span bridge and half-scale for the continuous system were modelled considering the appropriate boundary symmetry conditions. The focus was on (i) effects of mesh size of blast pressure, (ii) different amounts of explosives, (iii) different reinforcement ratios; and (iv) distance of the slab from the explosives. However, it is not mentioned how the mesh size of the considered RC slab may affect its dynamic response. A similar note is observed for the work of Kiger et al. [25].

Alañón et al. [13] investigated the mesh size sensitivity of 15 FEMs in reproducing the damage on an RC slab where (i) only the longitudinal half of the slab was considered due to symmetry, and (ii) reinforcing steel mesh size was fixed in all 15 FEMs. Alañón et al. [13] conducted a mesh sensitivity study examining variable mesh sizes ranging from coarse to fine for their FEM simulations. The authors claimed that they identified optimum mesh size for each loading condition based on the internal energy extent, which progressed to the probable slab damage. It should be pointed out that both Alañón et al. [13] and other studies such as Castedo et al. [14] took fixed lengths (same mesh size) for reinforcing steel bars with a different mesh size for the concrete; however, separate mesh refinement studies for the reinforcement and concrete should be performed. That is because the reinforcement and concrete are constructed independently of each other, especially when constraint methods are used.

Maazoun et al. [15] evaluated the effects of different material models of the concrete, hourglass energy, and erosion values on the dynamic response RC hollow-core slabs; the evaluation was in terms of the use and accuracy of finite element simulations by means of LS-DYNA. It is claimed that the accuracy of FEM is related to correctly predicting the maximum defection and crack distribution compared with experimental results. Other parameters such as hourglass energy, erosion value, and material models are considered only to evaluate their effects on the structural response of the considered structure. It can be seen that convergence/sensitivity issues and model stabilities were not mentioned.

3.2 Numerical stability

Numerical stability is a fundamental requirement of FEMs; it is a key to an accurate simulation. In the case of blast load and because of the nature of the load, the stability of the numerical model may relate to system (structure) energy. For instance, a numerical model can be considered as an accurate simulation if the energy ratio is equal to 1.0, or ranges between +/- 0.01 (LS-DYNA 2017 [21]).

Also, the total energy of the structure can be used as a measure of FEM quality. Typically, it is thought that the increase of the total energy over the initial energy should be less than 10%, and the ratio of the hourglass energy to the total energy be less than 10% (LS-DYNA 2017 [21]). It is important to mention that hourglass modes are nonphysical deformation modes of the finite element mesh that produce zero strain and no stress. Hourglass modes occur only in under-integrated (single integration point) solid, shell elements.

Another parameter that can control the model stability in the case of blast load is erosion. The erosion option represents a numerical way to delete the elements which satisfy given criteria. The failure criteria should be determined based on a sensitivity study because erosion is not a physical parameter. The add-erosion option provides a way of including the cracking of the concrete in the numerical model. This option can also be applied to constitutive models that already include other failure/erosion criteria (LS-DYNA 2017 [21]).

Maazoun et al. [15] considered parameters such as hourglass energy, erosion value, and the material models only to evaluate their effects on the structural dynamic response; however, these parameters (hourglass energy and erosion) are not considered for different mesh sizes in their study.

4. Optimum modelling approach

It should be pointed out that the main goal of a numerical simulation is to validate the simulation by matching experimental results in terms of deflections and prediction of damage/crack pattern and general structural behaviour. However, different aspects of the fundamental mechanics of FEM should be considered. For instance, in Tai et al. [3], the discussion was mainly focused on the meshing of blast pressure, size of explosive charges, and its distance from the slab; the reinforcement amount is also considered. However, the effects of mesh size of the slab and reinforcement were not considered. Also, Alañón et al. [13] explored the reliability of finite element simulations of an RC slab under a certain level of explosion; the focus was on considering the mesh size sensitivity on the RC slab behaviour in terms of damage pattern (which compared to experimental results) and internal energy history. For all their FEM simulations, Alañón et al. [13] studied the effects of mesh size on the numerical modelling efficiency. While the concrete mesh size is only considered, steel bar mesh-size refinement and their correlation with the concrete mesh sizes are not mentioned. In Castedo et al. [14], the major validation was in terms of damage pattern. As stated above, the effects of different model settings on the results are rarely reported, making it necessary to do more numerical studies to understand the complexity of the dynamic response of reinforced concrete structures. However, it is never mentioned how they have dealt with convergence/sensitivity and stability issues as they are significant keys to the accuracy of any numerical model. Maazoun et al. [15] described how to develop a numerical model to predict the maximum deflection and crack distribution, but they did not mention how they have dealt with convergence/sensitivity issues and model stabilities. They stated that the accuracy of FEM is related to correctly predicting the maximum defection and crack distribution as compared with experimental results. Other parameters such as hourglass energy, erosion value, and the material models are only considered to evaluate their effects on the structural response of the considered structure. Wenjiao et al. [16] conducted a series of numerical studies aiming to get dynamic data of the cracked RC slabs under blast loading. Constraints of steel and concrete are not clearly stated in the paper. Dynamic increase factor is included. Bond slips between steel bar and concrete are assumed to be perfect, which is not representative of the actual behaviour. Hence, requirements for more robust and accurate numerical algorithms are needed. Table 1 summarizes the consistency of different references considered in this paper with the effective FEM procedure for RC panels subjected to blast load. The table shows that a limited focus is given to verifying and validating the finite element models. These two tasks (verification and validation) are the essential steps in the development of reliable models that can safely be used to predict the structural performance when there is a change in the geometry, boundary conditions, and/or material properties.

Table 1. Consistency of Previous Studies with the Effective FEM Procedure for RC panels Subjected to Blast Load

	Fundamental Steps of Developing FEM for RC Panels under Blast Loading					**FEM Verification and Validation**	
	Approach	**Geometry &B.C**	**Materials**	**Contact**	**Blast Load**	**Verification**	**Validation**
Abdel-Mooty et al.[10]	√	√	√	Not Available	√	Not Available	Partial Validation
Alañón et al. [13]	√	√	√	√	√	Partial Verification	Not Available
Castedo et al. [14]	√	√	√	Not Available	√	Not Available	Partial Validation
Galuta & Regig [12]	√	√	Not Available	Not Available	√	Partial Verification	Not Available
Maazoun et al. [15]	√	√	√		√	Not Available	Partial Validation
Mosalam & Mosallam [16,18]	√	√	√	Not Available	√	Not Available	Partial Validation
Schwer [26]	√	√	√	√	√	Partial Verification	Not Available
Tai et al. [3]	√	√	√	√	√	Partial Verification	Partial Validation
Wenjiao et al. [16]	√	√	√	√	√	Partial Verification	Partial Validation
Zhou et al.[5]	√	√	√	√	√	Partial Verification	Partial Validation

The previous section shows that numerical models can be used to explore the feasibility of RC structural behaviour concepts prior to explosive testing and to model more complex structures affected by blast loads. However, using numerical models presents a set of challenges such as (i) selection of a suitable problem-specific mesh, (ii) ability to examine the stability of the solution procedure, and (iii) possible sources of errors based on modelling assumptions. The optimum modelling approach can be reached by considering the follows:

1) Selection of the appropriate element type for numerical analysis is essential in the finite element method (FEM). The selection is based on the desired level of accuracy and the time cost associated with each numerical method [5]. The compatibility of the selected element with the constitutive material model used in the numerical analysis procedure is also very essential.

2) Separate mesh refinement studies for the reinforcement and concrete should be performed when the constraint methods are used to model the reinforcement and concrete (that is because they are constructed independently of each other in constraint methods). Also, Mesh convergence should be conducted where everything else in the model is checked to be working well (material models, contact, energy, etc.).

3) Mesh sensitivity analysis needs to be conducted to yield accurate results with minimal computational effort, especially when making blind predictions of the behaviour of structures.

4) From the physical point of view, the concrete fracture progress zone (CFPZ), the area in which all the cracks are produced, is assumed to occur within one element of the mesh for the concrete numerical model. If the mesh size is greater than the CFPZ, the fracture energy can be dissipated completely. In contrast, if the mesh size is smaller than the CFPZ, the fracture energy cannot be dissipated, overestimating the real damage [14].

5) Mesh of the blast load simulation should be finer than the mesh of the structure.

Mass scaling in CPU management time/time step could be the first step to verify the model in terms of time-consuming (CPU management).

6) Time step must be sufficiently small to provide solution stability.

7) Fundamental results of a simulation should be examined. Then the three categories of data must be examined to assess solution validity, including (i) global summations such as global energy, global force sum, global energy sum, and % of damping or kinetic energies to internal; (ii) nodal results such as displacements and summation of forces; and (iii) element results.

8) Energy should be small relative to internal energy

9) Contact energy and hourglass energy (a combination of hourglass force and displacement) are key tools to verify numerical model stability.

It should be pointed out that considerable efforts have been conducted in the area of blast loading response of RC elements such as RC columns [28-32]; the work efforts have two parts: experimental and theoretical. However, this paper aims to comprehensively review on existing modelling studies in the literature investigating the loading mechanisms, dynamic responses, and failure behaviors of RC panels subjected to blast loads. The work addresses the research gaps associated with using advanced finite element modelling as compared to test results.

5. Conclusions

The main goal of a numerical simulation is to match experimental results in terms of deflection and predict crack/damage patterns considering that the numerical model simulation should satisfy finite element requirements. In many applications that used LS-DYNA, attention was not paid to satisfy the requirements of numerical models in terms of stability and verifications. On the other hand, it is essential to understand the limitation of the code/software and other issues that cannot be controlled. This will help in the verification process and selection of appropriate options in order to provide the best model. For instance, using LS-DYNA, the following cannot be controlled: (i) failure equations, strain rate sensitivity models, and/or post-failure and damage models in a material model; (ii) search algorithm in the contact; (ii) formation in an element; (iii) time integration scheme in implicit and explicit schemes; and (iv) advection algorithm in ALE. It should be pointed out that validation and verification decisions need to be based as much as possible on quantitative criteria that are unambiguous and mathematically precise [27].

In some applications, experimental/field data are not available to compare results of numerical simulation; in such case, sensitivity analysis of the numerical simulation must be conducted, including: (i) contact type and contact parameters; (ii) hourglass type; (iii) element type; (iv) time step scale factor in explicit and time step size in implicit; (v) material properties perturbation; (vi) number integration points through-thickness; (vii) type and details of connections; and (viii) constraints assumptions.

The use of three-dimensional half or full-scale finite element modelling (with or without structural symmetry) provides a comprehensive understanding of the structural performance. Verification and validation of the FEM would enable a reliable prediction and simulation of the failure mechanism and the damaged patterns of RC panels when subjected to blast load. Generalization of the FEM applications on different boundaries, dimensions, and reinforcement details would be possible if the numerical performance and the validation consistently give stable and acceptable results.

6. References

[1] Lu Y. Modelling of concrete structures subjected to shock and blast loading: An overview and some recent studies. Structural Engineering and Mechanics. 2009;32(2):235-249.
[2] Zhang C, Gholipour G, Mousavi AA. Blast loads induced responses of RC structural members: State-of-the-art review. Composites Part B: Engineering. 2020;195:108066.
[3] Tai YS, Chu TL, Hu HT, Wu JY. Dynamic response of a reinforced concrete slab subjected to air blast load. Theoretical and applied fracture mechanics. 2011;56(3):140-147.
[4] Kei C. Effects of blast loading on seismically detailed reinforced concrete columns. Master of Applied Science in Civil Engineering, Carleton University Ottawa, Ontario. 2014.
[5] Zhou XQ, Kuznetsov VA, Hao H, Waschl J. Numerical prediction of concrete slab response to blast loading. International Journal of Impact Engineering. 2008;35(10):1186-1200.
[6] Zienkiewicz OC, Cheung IK. Finite Element Method in Engineering Science. McGraw-Hill.1971.
[7] Gallagher RH. Finite element analysis: Fundamentals. Prentice-Hall, Englewood Cliffs, N.J..1975.
[8]. Burnett DS. Finite Element Analysis: From Concepts to Applications. Addison Wesley Publishing Company,1987.
[9] Bathe KJ. Finite Element Procedures. Prentice Hall PTR, 1995.
[10] Abdel-Mooty M, Alhayawei S, Issa M. Numerical evaluation of the performance of two-way RC panels under blast loads. Structures under Shock and Impact XIII Conference. June 2014. p.13.
[11] Razaqpur AG, Tolba A, Contestabile E. Blast loading response of reinforced concrete panels reinforced with externally bonded GFRP laminates. Composites Part B: Engineering. 2007;38(5-6):535-546.
[12] Galuta E, Regig W. Numerical simulations of RC panels subjected to high speed projectile-erosion selection in AUTODYN-3D code. International Journal of Innovative Science, Engineering & Technology. 2017; 4(8):25-30.
[13] Alañón A, Cerro-Prada E, Vázquez-Gallo MJ, Santos AP. Mesh size effect on finite-element modeling of blast-loaded reinforced concrete slab. Engineering with Computers. 2018;34(4):649-658.
[14] Castedo R, Segarra P, Alañón A, Lopez L, Santos A, Sanchidrian J. Air blast resistance of full-scale slabs with different compositions: Numerical modelling and field validation. International Journal of Impact Engineering. 2015; 86:145-156.
[15] Maazounab A, Vantommea J, Matthysb S. Damage assessment of hollow core reinforced and prestressed concrete slabs subjected to blast loading. X International Conference on Structural Dynamics. Procedia Engineering. 2017;199: 2476–2481.
[16] Wenjiao Z, Kong X, Qu Y, Zhao Q. Numerical simulation of cracked reinforced concrete slabs subjected to blast loading. Civil Engineering Journal. 2018;4(2):320-333.
[17] Mosallam A, Mosalamb K. Nonlinear transient analysis of reinforced concrete slabs subjected to blast loading and retrofitted with CFRP composites. Composites Part B. 2001;32:623–636.
[18] Mosallam A, Mosalamb K. Strengthening of two-way concrete slabs with FRP composite laminates. Construction and Building Materials. 2003;17:43–54.
[19] Mays GC, Hetherington JG, Rose TA. Resistance-deflection functions for concrete wall panels with openings. Journal of Structural Engineering. 1998;124(5):579-587.
[20] Mays GC, Hetherington JG, Rose TA. Response to blast loading of concrete wall panels with openings. Journal of structural Engineering. 1999;125(12):1448-1450.
[21] Livermore Software Technology Corp. (LSTC). LS-DYNA keyword user's manual, Livermore, CA. 2017.
[22] Tabatabaei Z, Volz J. A comparison between three different blast methods in LS_DYNA: LBE, MM-ALE, coupling of LBE and MM-ALE. 12th International LS-DYNA® Users Conference. 2012.
[23] Shukla P, Desai A, Modhera C. The current practice of analaysis of reinforced concrete panels subjected to balst loading. World Academy of Science, Engineering and Technology. International Journal of Structural and Constructions Engineering. 2018;12(10).
[24] LSTC. 2017b. LS-DYNA, Keyword User's Manual: Volume II, Material Models. Livermore Software Technology Corporation (LSTC), Livermore, California.
[25] Kiger S, Salim H, Ibrahim A. Bridge Vulnerability Assessment and Mitigation against Explosions. Final report from Midwest Transportation Consortium, Iowa State University. 2010.
[26] Schwer L. Modelling rebar: the forgotten sister in reinforced concrete modelling. 13th International LS-DYNA® Users Conference. Jun. 2014.
[27] Tabiei A. Verification and Validation in LS-DYNA. WWW.LSDYNA.ONLINE.COM. 2021.

[28] Zhou Y, Yang J, Luo X, Hwang HJ, Chen H, Sun J, Yi W, Kang SM. Pendulum impact loading tests of precast concrete columns with various column base connections. Engineering Structures. 2022;252:113736.
[29] Moussa AM, Fahmy MF, Wu Z. Innovative resilient system of precast segmental RC hollow bridge columns. Engineering Structures. 2021;229:111555.
[30] Gholipour G, Zhang C, Mousavi AA. Nonlinear numerical analysis and progressive damage assessment of a cable-stayed bridge pier subjected to ship collision. Marine Structures. 2020;69:102662.
[31] Dua A, Braimah A, Kumar M. Experimental and numerical investigation of rectangular reinforced concrete columns under contact explosion effects. Engineering Structures. 2020;205:109891.
[32] Li H, Chen W, Hao H. Dynamic response of precast concrete beam with wet connection subjected to impact loads. Engineering Structures. 2019;191:247-263.