On the Assessment of Existing RC Structures by Virtue of Nonlinear FEM: Possibilities and Challenges

Jiabin Li
Associate Professor, Dr.-Ing.
Oslo and Akershus University College
Pilestredet 35, N-0130 Oslo, Norway
Mailto: lijiabin6598@hotmail.com
Professor, Dr.-Ing.
KU Leuven Campus Bruges
Spoorwegstraat 12, 8200 Bruges, Belgium
Mailto: jiabin.li@kuleuven.be

Glenn Strypsteen
PhD candidate
KU Leuven Campus Bruges
Spoorwegstraat 12, 8200 Bruges, Belgium
Mailto: glenn.strypsteen@kuleuven.be

Mahdi Kioumarsi
Associate Professor, PhD.
Oslo and Akershus University College
Pilestredet 35, N-0130 Oslo, Norway
Mailto: Mahdi.Kioumarsi@hioa.no

ABSTRACT

This paper discusses the possibility and challenges of the assessment of the structural integrity and safety of existing reinforced concrete (RC) structures by virtue of the nonlinear finite element method (FEM). The possibility and viability of applying this tool to predict the mechanical behaviour of existing RC structures are discussed through two sample applications, which deal with the estimation of the actual loading capacities of a RC column-slab joint strengthened by steel plates and a RC parapet element with inadequate transverse reinforcements according to current design codes. The numerical results demonstrate that when properly used, the nonlinear FEM is able to effectively remedy the inadequacy of conventional design and analysis procedures. Some remaining challenges for the utilisation of nonlinear FEM to assess existing RC structures are then discussed.

Key words: Existing reinforced concrete (RC) structures, structural safety, remaining loading capacity, finite element method (FEM), microplane model

1. INTRODUCTION

The assessment of the real structural integrity and safety of existing reinforced concrete (RC) structures is becoming a more and more important issue for civil engineers. The actual load carrying capacities or the safety factor of existing RC structures need to be known owing to the following reasons:

- Increased loads on existing structures (particularly on bridges due to the increase of the traffic load);
- Material deteriorations such as the corrosion of reinforcing steel and the alkali silica reaction (ASR) damage in concrete;
- The generation of new design codes;
- Pre-existing damages in the structures (e.g. premature cracking in concrete) as well as structural faults (e.g. design and construction error).

Nonlinear finite element method (FEM) provides an efficient tool for the realistic assessment of the actual behaviour and failure of existing RC structures. Since the pioneering work of Ngo and Scoreless [1], numerous efforts have been made in this field, mainly in constitutive modelling of the material behaviour and development of sophisticated analysis algorithms [2]. Given these continuous progresses, nonlinear FEM simulation of RC structures has matured to a significantly high level. Compared to three decades ago, many problems encountered in the engineering practice can be better investigated today by using of this tool. In addition, there is no limit to the applications of this method.

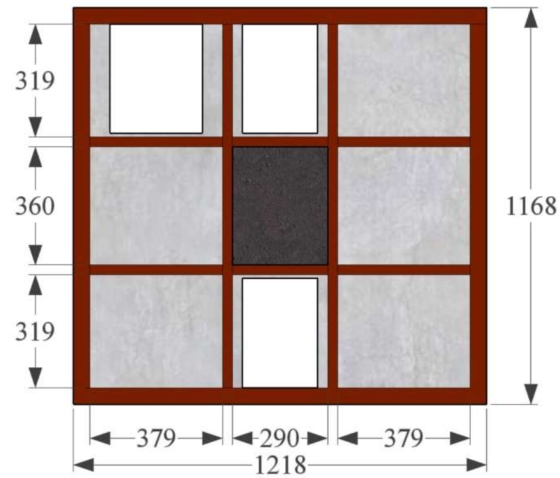
A number of successful applications of nonlinear FEM in the assessment of existing RC structures and bridges including some very complex structural systems have been reported in the literature [3-6]. By virtue of this method, the mechanical responses and failure loads of the investigated structures have been better understood. This has led to more rational and economical strategies for the structures. In the past several years, the first author of this paper has successfully completed a series of structural assessments of various types of existing RC structures through the use of nonlinear FEM, with significant economical and environmental benefits. These benefits have been achieved by avoiding unnecessary material consumptions and construction activities. Two sample applications are presented below.

2. TWO SAMPLE APPLICATIONS

2.1 Prediction of loading capacity of a RC column-slab connection

In one building in the Public Hospital in Graz, Austria, steel profiles were used to strengthen the RC column-slab connections to benefit the load transfer among different components. The concept of the construction is illustrated in Figure 1, in which a typical type of the column-slab connection is shown. In this element, the RC column has a cross section of 300 × 400 mm^2. The thickness of the concrete cover is 30 mm. The inner steel profile has a rectangular cross section (width: 30 mm, height: 150 mm) while the outer steel profile is a U-shaped profile, which is U120/55/6 mm according to DIN 1026-1. No shear connectors were used between the steel profiles and the column concrete as well as between the steel profiles and the slab concrete. Outside the column-slab connection region, unreinforced concrete was filled into the steel profiles. The basic design consideration is that the slab loads transfer firstly to the steel profiles then from the steel profiles further to the RC column.

A notable (or perhaps less efficient) point of this design is that the inner steel profiles are placed (more or less) directly on the RC column concrete cover, see Figure 2.



(a) Three-dimensional (3D) view (b) Top view

Figure 1 Construction concept of a typical column-slab connection strengthened by steel profiles (Units: mm. Note: There are openings in the slab)

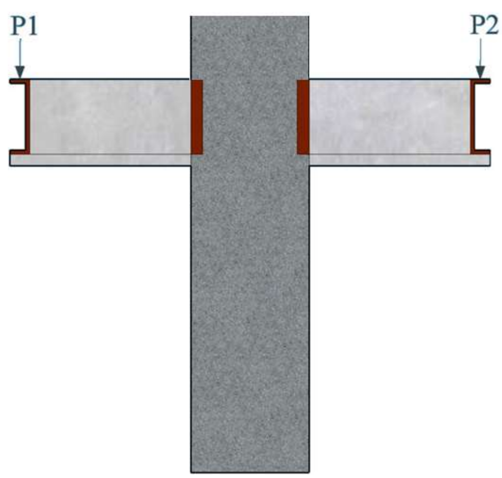
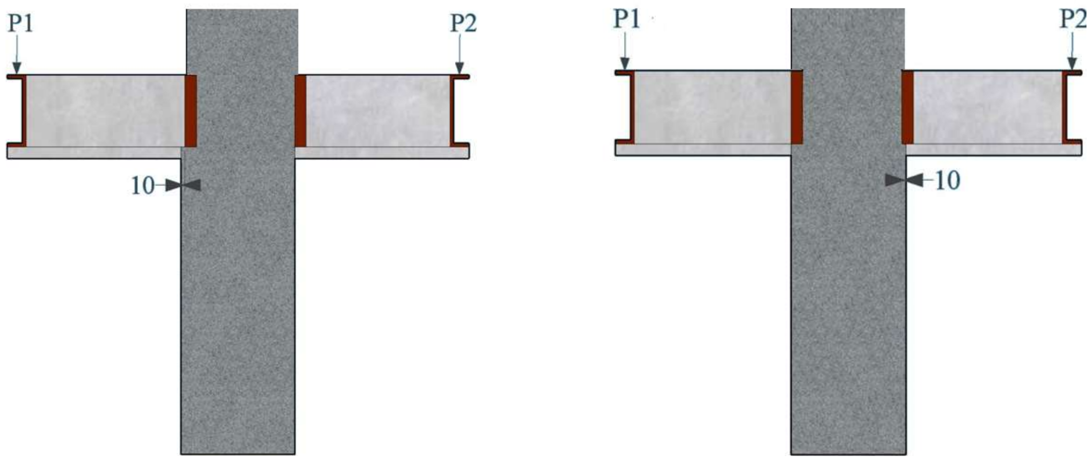


Figure 2 Designed (favorable) positions of steel profile with reference to RC column

After the construction, however, it was found that the inner steel profiles of many column-slab connections are located in rather unfavorable positions. Some typical cases are illustrated in Figure 3. In the case shown in Figure 3(a), the left inner steel profile is supported on the concrete cover with a distance of 10 mm from the left edge of the steel profile to that of the column concrete cover; while the right inner steel profile is placed outside the column concrete cover (!). In the case in Figure 3(b), the left steel profile is precisely placed on the column concrete cover while the right profile is only partially (10 mm!) supported on the column cover.



(a) Unfavorable position 1 (b) Unfavorable position 2

Figure 3 Typical actual (unfavorable) positions of steel profile with reference to RC column

There are several types of column-slab connections in the building. The differences between various connection types include the structural dimensions, the column and the slab concrete strength grades as well as the service load level. A simplified engineering analysis, in which the slab loads are assumed to be completely exerted on the column concrete cover, indicated that under the service slab load, a high splitting stress occurs in the column cover concrete and longitudinal cracks formulate in the column cover concrete. Thus, a common opinion had been that when the loads on the slab increase to the ultimate load level, a spalling of the column concrete cover would occur and the steel profiles would then fall to the ground. Thus, a rehabilitation of the structure had been considered to be necessary.

The building had been in service for several years. However, on-site monitoring did not detect any visible longitudinal cracks in the column cover concrete. This is inconsistent with the results of the simplified analysis since longitudinal cracking in the column cover is expected according to the analysis. To better understand the real behaviour of the structure, more sophisticated analysis is obviously required. Three-dimensional (3D) nonlinear FEM simulations were thus performed to investigate the responses of the structure under ultimate design load.

Since the objective of this section is to demonstrate the possibility and viability of the use of nonlinear FEM to predict the load carrying capacity and failure behaviour of exiting RC structures, it is not the intention of the authors to let the readers repeat these simulations. Thus, a very detailed description of the numerical models and procedures is not given here. The modelling details of the column-slab connections can be found in [7-9]. In this part, only the two most important points in the numerical simulations of the structures are highlighted, which are the constitutive modelling of the column concrete and slab concretes, as well as the simulation of the contact between the steel profiles and concretes. The microplane model M4L for concrete developed in [10-12] was employed to simulate the mechanical behaviour of the concretes. This model has been proven to be able to realistically capture the uniaxial, biaxial and triaxial mechanical behaviour of concrete. More information about the numerical performance of the model for simulating the concrete properties under different kinds of stress states are presented in [11-12]. The discrete contact spring model [9] is employed to simulate the interface behaviour between the steel profiles and the column as well as slab concretes. The constitutive behaviour of the steel material is described by the von Mises yield criterion with isotropic strain hardening and an associated flow rule. Both the concretes and the steel profiles are discretised into 8-node solid elements with $2\times2\times2$ integration points. Figure 4 presents the 3D finite element mesh for a typical connection system.

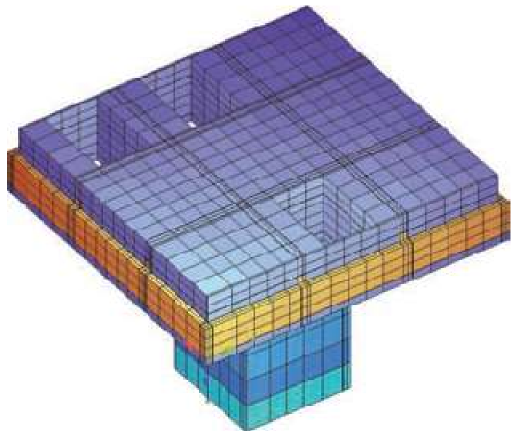


Figure 4 3D finite element mesh of a typical column-slab connection

The detailed numerical simulation results for various connection systems are given in [7], which led to the following interesting findings:

- Under the service slab load, the tensile splitting stress in the column cover concrete is significantly smaller than the concrete tensile strength and thus no cracking in the concrete occurs. This is consistent with the on-site observations;
- The load carrying capacities of the connections are significant higher than that derived by the simplified analysis. The predicted maximum slab loads of the connections are about 2.6 times the service loads, indicating that the load bearing capacities of the connections are adequate according to Eurocode 2 [13];
- For all the investigated connections, the failure was due to the yielding of the inner steel profiles rather than the spalling of the column cover concrete. A falling of the steel profile to the ground at the maximum slab load was not observed in the numerical simulations.

With the guidance of the numerical simulation results, full-scale tests on two types of the connections were performed in the laboratory [14]. The test results were in very good agreement with that predicted by the numerical model. The failure of the tested connections were due to the yielding of the inner steel profiles rather than the spalling of the column cover concrete, and the load carrying capacities of the connection were adequate. More information about the test results and their analysis can be found in [14]. After the tests, numerical analysis of the test specimens was also done using the measured material properties of the column and slab concretes. The predicted deformation behaviour and the failure loads of the connections were very close to the test observations. Figure 5 shows the simulated load-deformation curve of a connection.

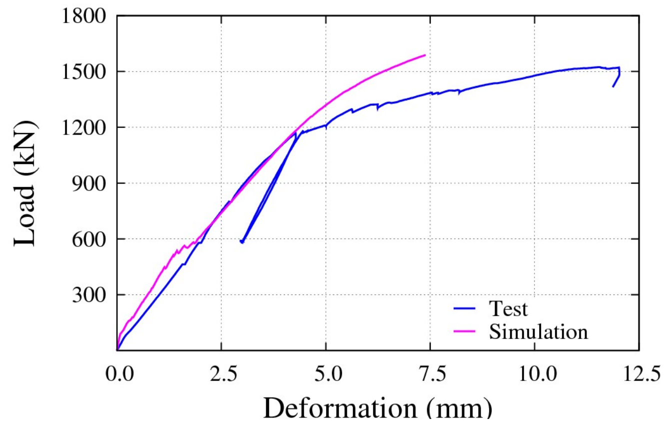


Figure 5 Comparison of the load-deformation curve from numerical simulation and test

Because the numerical model was successfully verified by the test data, it is thus possible to investigate the load transfer mechanism in the column-slab system to better understand the actual response of the structure. The analysis results were presented in [9]. It was found that the friction between the inner steel profiles and the column concrete has a significant contribution to the load bearing capacities of the connections. This is because when there is relative movement between the inner steel profile and column concrete under the slab (bending) load, the friction between the surfaces of the two materials activates and this friction transfers a part of the load from the slab to the column core concrete (not the column cover concrete!) . This explains why the actual load carrying capacities of the column-slab connections are much higher than that estimated from the simple engineering analysis.

Further, the load carrying capacities of the rest of the connection systems were predicted by using of the validated numerical model. It was found that all the connections have adequate load bearing capacities. Therefore, no retrofit of the structure is required. This finally led to a significant cost because of the avoidance of unnecessary material consumptions as well as construction activities.

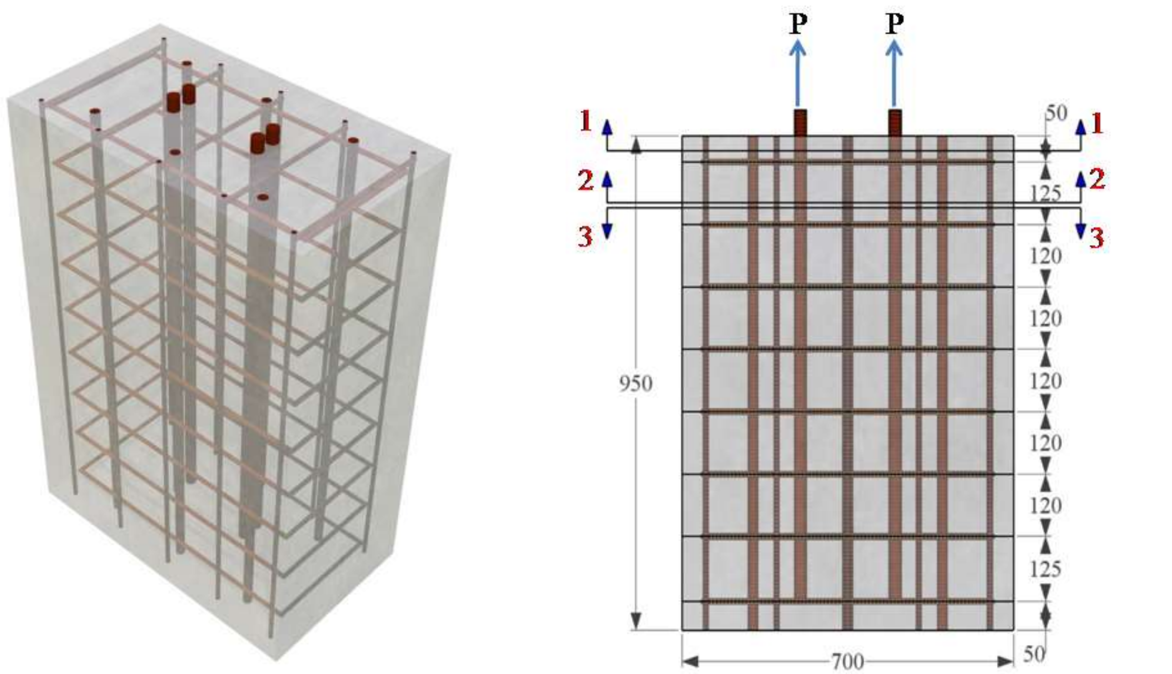
2.2 Estimation of maximum load of a parapet with low transverse reinforcement

This example deals with the load bearing capacity of a RC parapet element with low amount (insufficient) transverse reinforcements according to the design codes. In one garage at the Berlin-Brandenburg International Airport, RC parapet elements were constructed to resist the force of the bracing system. The longitudinal reinforcements in these elements were overlapped with the connecting reinforcements in the foundation. The lapped length of the reinforcements met the requirements of the German design code DIN 1045-1 [15]. However, in some built elements, the amount of the transverse reinforcements for resisting the possible splitting tensile force in the RC parapet element is too low in comparison to that required by DIN 1045-1. The available reinforcements were found to be insufficient to bear the splitting tensile force determined by the engineering model recommended in DIN 1045-1 [15]. Therefore, a post-installation of additional transverse reinforcements had been considered to be necessary. However, this is not an optimal choice since it would impose the structure into an unfortunate condition due to the post installation of additional transverse reinforcements. On the other hand, since the engineering model in DIN 1045-1 is generally conservative, the use of this model might be inadequate to determine the real load carrying capacity of the parapet elements.

To investigate the real load carrying capacity of the parapet elements, 3D nonlinear FEM simulations were carried out [16]. A representative element was computationally studied. To reduce the computational expense, only a strip of the parapet element (cross section: 400 × 700 mm^2, height: 950 mm) was simulated. The 3D view and the front view of the modelled parapet element are shown in Figure 6(a) and 6(b), respectively. The longitudinal reinforcements in the modelled parapet element are shown in Figure 7 (a), which include three groups:

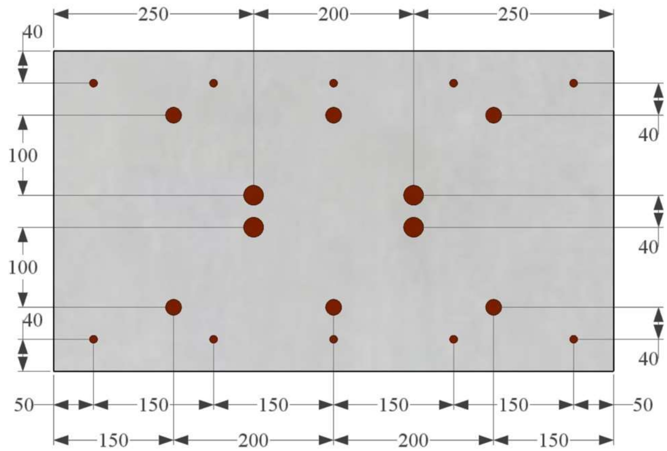
- Group 1: 10 Φ10 reinforcements in the outer edge of the cross section;
- Group 2: 4 Φ 25 reinforcements in the middle of the cross section; and
- Group 3: 6 Φ 20 reinforcements between the above two groups of reinforcements.

The transverse reinforcements have a diameter of 10 mm. However, it should be noted that the reinforcements of the top layer are different from that of the other layers, as shown in Figure 7(b) and 7(c), respectively. More details of the modelled parapet element, such as the material properties of the concrete and the reinforcements can be found in [16].

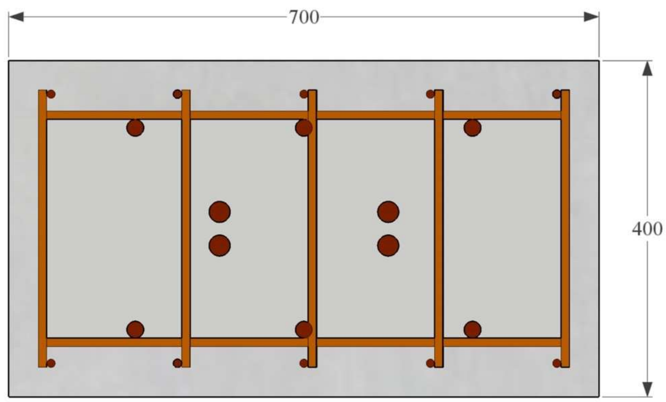


(a) 3D view (b) Front view

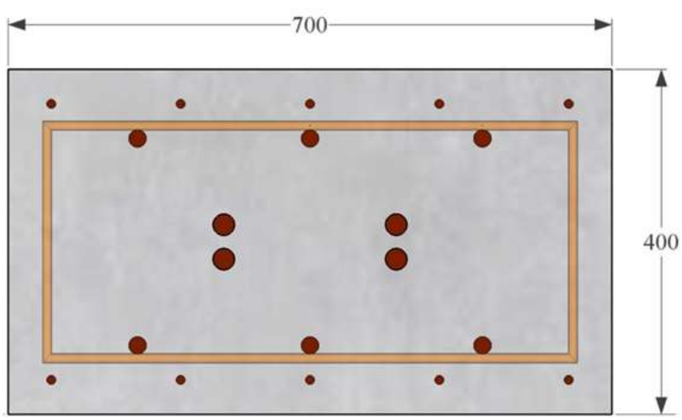
Figure 6 Geometry of the modelled parapet element without additional transverse reinforcement (Units: mm)[16]



(a) Section 1-1



(b) Section 2-2



(c) Section 3-3

Figure 7 Details of the reinforcements in the modelled parapet element without additional transverse reinforcement (Units: mm) [16]

The microplane model M4L for concrete [7-9] was again used to simulate the concrete in the parapet element. All the reinforcements in the element were modelled as an elastic-perfect plastic material described by its modulus of elasticity E_s and the yield strength f_y. The bond-slip behaviour between the concrete and the reinforcements plays an important role in the simulation of the element. The bond-slip model in MC90 [17] was employed in the numerical model. The concrete was discretised into 8-node solid elements with 2×2×2 integration points, while the reinforcing steel was simulated with 2-node truss element. More details about numerical modelling, such as the material parameters, the finite element mesh and the solution algorithm etc can be found in [16].

The numerical results indicated that the failure of the simulated RC parapet element was caused by the yielding of the four axially tensioned reinforcements with a diameter of 25 mm; while an inclined splitting failure in the concrete was not detected. This is due to the fact that the four load-introducing longitudinal reinforcements (Φ 25) have a relatively large side concrete cover. Owing to the thick concrete cover, the low amount of the loading reinforcements and the actual transfer of the load between the reinforcements and the concrete, the splitting failure of the concrete in the element was delayed or prevented. Depending on the bond strength between the concrete and the steel reinforcements, the failure of the parapet element was caused by the pull-out failure or yielding of the four 25 mm diameter longitudinal reinforcements, prior to the tensile splitting failure of the concrete. The predicted failure load, which is in fact the yielding load of the four loading longitudinal reinforcements, is approximately 1.80 times the design load, indicating the parapet element is still on the safe side. An extensive parameter study through varying the tensile strength of the concrete and the bond strength between the concrete and the reinforcing steel was also conducted, yielding a better understanding of the load carrying behaviour of the parapet element. The numerical results and their discussions are given in [16, 18].

On the basis of the numerical simulation results, it was found that no post-installation of additional transverse reinforcement would be necessary.

3. SOME REMAINING CHALLGENCES

From the above two samples in this paper as well as some other successful applications reported in the literature, it can be concluded that when properly used, nonlinear FEM is able to serve as a very useful tool in the assessment of the real structural integrity and safety of existing RC structures. Despite that, some rather big challenges still remain in the practical application of this tool, especially when the simulations of RC structures are done in engineering offices with commercial finite element packages. Some of these challenges are discussed in the following.

3.1 Numerical Accuracy

The question on how accurate are the numerical results or how confident is the engineer in the calculation results is commonly asked when nonlinear FEM simulations of RC structures are used in engineering practice, in particular when the mechanical behaviour of the structure is not yet fully understood and no test data are available. Numerous excellent simulations of the load-deformation curves of different RC structures have been reported in the literature; however, many results are not adequate if compared with the test observations on the crack propagation, the reinforcement strain and the failure mode [19].

The robustness and performance of the concrete material model often governs the accuracy of the simulation results. A large amount of concrete constitutive models have been developed. Unfortunately, none of existing models is able to realistically represent all important material behaviour of concrete subjected to general loading conditions, since the modelling of concrete is a notoriously hard task. Most of the models can only yield case-dependent success and there are always arguments on the objectivity of the numerical simulation results. The use of nonlinear FEM simulations incorporating insufficient concrete material models to RC structures in engineering practice can result in very big risk of structural failure, if the real load bearing capacity is overestimated by the numerical model. The formulation of a robust and versatile constitutive model of broad applicability and capability is still a big challenge.

On the other hand, a RC structure consists of different constituents: concrete, reinforcement and sometimes also structural steel. The nonlinear actions of the individual constituents are in some cases an important cause of the responses of the structure. Therefore, material models for these composite actions, such as the bond-slip between reinforcement and concrete, the aggregate interlock (shear friction) in cracked concrete when lateral compression exists, and the dowel action of steel reinforcement as well as the bond-friction between concrete and structural steel, are required to precisely represent the behaviour of the structure. Unfortunately, adequate models for describing these actions are not yet available. Some existing commercial FE packages provide options for simulating some types of actions, e.g. the interface behaviour between concrete and structural steel. However, they generally belong to quite simplified solutions and the scope of application is still rather limited. How to realistically simulate these interaction behaviours and how to select the model parameters in the structural simulations are also big challenging problems and more efforts in this field are necessary.

It is well known that concrete exhibits softening behaviour under many types of loadings. When this material behaviour is included, a problem related to the finite element mesh, known as mesh dependency, frequently occurs, including the dependency on element size and the dependency on mesh pattern. The former refers to the size of the localized zone decreases as the element size decreases. The crack band model is often used in commercial packages to minimise the influence of the element size on the numerical results for tensile dominated failures. However, when the crack band model is used, the simulated cracks tend to develop parallel to the element mesh lines, that is, a dependency of the mesh pattern. For this reason, the crack band model is generally difficult to use unless the crack pattern of the investigated problem is known in advance, which is, however, often not the case for practical applications.

Smeared crack models are commonly implemented in commercial finite element packages for simulating RC structures. However, when the softening branch of the stress-strain curve of concrete is adjusted to keep the fracture energy constant according to the concept of the crack band model, the simulated cracks often occur in multiple elements instead of being localized. This is not correct since the fracture energy and the stress-strain curve are given assuming that the crack is localized in one element. The solution is acceptable only when the used element size coincides with the macroscopic crack spacing of the actual structure.

Other techniques such as the non-local approaches and the gradient type methods might be more suitable for avoiding the mesh dependency problems. However, these techniques are not yet popular and they are absence in most commercial finite element packages. Generally, the mesh dependency in nonlinear FEM simulations of RC structures has not been well solved. Simple and efficient approaches to avoid the mesh dependency are not available. At present, when nonlinear FEM simulations are used for predicting the behaviour of existing RC structures in practice, it is often recommended that mesh sensitivity tests should be performed to validate the finite element mesh of the model. Different mesh cases with different element sizes and their effects on the numerical results should be assessed [20]. In the case of significant mesh sensitivity, the numerical model should be considered as not objective.

3.2 Numerical Efficiency

When nonlinear FEM simulations are used in engineering offices to investigate the integrity of existing RC structures in practice, the efficiency of the simulations can be even more important than the accuracy. This is because the work in engineering offices has to be concerned with time, money and personnel. In such a case, deadlines are very important and the task must be completed on time with available resources.

Compared to conventional specimens in the laboratory, existing RC structures in the engineering practice are usually of much larger scale. On the other hand, triaxial stress states frequently occur in most RC structures. Hence, 3D modelling needs to be used to achieve more realistic simulation results. Only in some special cases, the problems can be simplified into one- or two-dimensional problems. In the 3D numerical model of a real RC structure, an enormous number of finite elements and degrees of freedoms are often included. Moreover, to accurately represent the structural behaviour, advanced material models are often needed. Numerical experiences indicate that even with today's state-of-the-art computers, nonlinear FEM simulations incorporating sophisticated material models can become computationally intractable in case of large amounts of finite elements and/or complex stress states in structures. In most cases, numerical convergence problems lead to expensive computational cost. The computational expense greatly hampers the practical utlisation of nonlinear FEM simulations of existing RC structures and it can be regarded as one important obstacle.

High performance computers and parallel algorithm on a work station are helpful for reducing the computational time. However, these resources are usually not available in most design or consulting offices. New algorithms, such as multi-scale techniques and parallel algorithm on single or separated personal computer are useful for improving the efficiency of the computer simulations in engineering offices.

3.3 Safety Format

In addition to the accuracy and the efficiency of the computer simulations, the reliability of using the simulation results within codes of practice framework is another important issue. In other words, how to reliably use the simulation results in conjunction with codes of practice? A sound safety format for the numerical analysis is another challenge related to the use of nonlinear FEM in assessing existing structures in engineering practice.

The usual design condition is written as

$$F_{\mathrm{d}} < R_{\mathrm{d}} \tag{1}$$

where, F_{d} is the design actions and R_{d} is the corresponding design resistance. F_{d} is predicted with the mean values of the material parameters while R_{d} should be assessed using the design values according to the requirement of the semi-probabilistic approach. Nevertheless, as discussed in [21-22], the mean values of the material parameters should be used in the nonlinear analysis. When the test data are compared with the numerical results, the actual structural behaviour can be reproduced only when the mean values are used. The use of design values in the nonlinear FEM analysis result in an erroneous assessment of the structural behaviour, the deformability and the load bearing capacity. This causes some difficulties for evaluating the reliability when applying the nonlinear simulation results within mostly design code of practice framework.

Several safety formats have been proposed to solve this inconsistency, for example, in [20, 22 23]. However, a sound safety format is still missing, especially when complex material models are used since they are calibrated with limited test data. Currently, the safety of the nonlinear FEM simulations is usually assessed based on the characteristic values of the material properties [24], which are also used in the two sample application presented in this paper. Obviously, there are still some inconsistencies with this methodology.

4. CONCLUSIONS AND REMARKS

This paper deals with the assessment of existing RC structures by virtue of nonlinear FEM. Two recent sample applications are presented to illustrate the possibility and viability of this tool. Some challenges of using nonlinear FEM in predicting the behaviour of existing RC structures in the engineering practice are also discussed. The following conclusions can be drawn from this study:

- Considerable progress has been achieved in nonlinear FEM simulation of RC structures. Together with the relentless increase of the computer power, this has made it possible to apply nonlinear FEM to assess existing RC structures in practice;
- When properly used, nonlinear FEM is able to provide a very efficient tool to rationally assess the real mechanical response and load carrying capacity of existing RC structures;
- In comparison to conventional analysis or design procedures in current codes of practice, Nonlinear FEM is able to give more realistic descriptions of the real behaviour and load carrying capacity of existing RC structures;
- In spite of many successful applications, nonlinear FEM should be used with caution in the assessment of existing RC structures, especially when used in engineering offices with commercial software;
- There are still several big challenges when nonlinear FEM is utilised to predict the behaviour of existing RC structures. They are mainly related with the numerical accuracy, numerical efficiency and safety format. Further developments in these fields will greatly increase the viability of this tool.

REFERENCES

1. Ngo D & Scordelis A C: “Finite Element Analysis of Reinforced Concrete Beams,” *Journal of American Concrete Institute*, Vol. 64, No.3, 1967, pp.152-163.
2. fib Task Group 4.4. *Practitioners’ Guide to Finite Element Modeling of Reinforced Concrete Structures*. Bulletin 45, International Federation for Structural Concrete (fib), 2008.
3. Mahalingam S & Sharma P: “The Use of Finite Element Analysis in Practical Concrete Design: Case Studies of Longholme Bridges and Other Assessments,” *Finite Element in Engineering and Science*, Rotterdam: A.A. Balkema, 1997, pp.93-106.
4. Vecchio F J: “Non-linear Finite Element Analysis of Reinforced Concrete: At the Crossroads?” *Structural Concrete*, Vol.2, No.4, 2001, pp. 201-212.

5. Vecchio F J: "Contribution of Nonlinear Finite-Element Analysis to Evaluation of Two Structural Concrete Failures," *ASCE- Journal of Performance and Construction Faculties*, Vol. 16, No.3, pp.110-115.
6. Maekawa K, Okamura H, Pimanmas A: "*Nonlinear Mechanics of Reinforced Concrete*," London: SPON Press, 2003.
7. Tue N V & Li J: "FE Berechung zur Ermittlung der Traglast von Stützen/Deckenknoten Bauwerk LKH, Graz West, " Technical Report, Institut für Betonbau, TU Graz, 2011.
8. Tue N V & Li J: "FE Berechnung zur Ermittlung der Traglast von Deckenknoten mit Pilztyp 7, 8 und 9 Bauwerk LKH Graz West, " Technical Report, Institut für Betonbau, TU Graz, 2012.
9. Tue N V, Li J, Turner K: "Advanced Numerical Simulations for Prediction of Bearing Capacity and Failure of Complex RC Structures," *Structural Engineering International*, Vol. 24, No.2, 2013, pp.211-216.
10. Li J: "*Development and Validation of a new Material Model for Concrete On the basis of Microplane Theory*," Graz: Verlag der TUG, 2011.
11. Tue N V, Li J, Caner F C: "Microplane Constitutive Model M4L for Concrete. I: Theory," *Computers & Structures*, Vol.128, 2013, pp.219-229.
12. Li J, Tue N V, Caner F C: "Microplane Constitutive Model M4L for Concrete. II: Calibration and Validation," *Computers & Structures*, Vol.128, 2013, pp.146-159.
13. European Committee for Standardization (CEN). Eurocode 2. Design of concrete structures, 2009.
14. Tue N V & Turner K: "Versuchsbericht zum Durchschlupfwiderstand von ausgewählten stahlpilzverstärkten Deckenknoten des LKH Graz West," Technical Report, Institut für Betonbau, TU Graz, 2012.
15. DIN 1045-1. Tragwerke aus Beton, Stahlbeton und Spannbeton – Teil 1: Bemessung und Konstruction, 2008.
16. Tue N V, Schneider H, Li J: "Tragfähigkeit der Brüstungen mit geringer Querbewehrung beim Parkhaus PNA des BBI, "Technical Report, Institut für Betonbau, TU Graz, 2011.
17. CEB-FIP. Model Code (MC) 90. Thomas Telford, 1993.
18. Tue N V & Li J: "Computer Simulations of RC Structures for Practice," *Structural Engineering International*, Vol. 24, No.2, 2013, pp.222-228.
19. Rokugo K & Uchida Y: "Trends in Numerical Analysis of Concrete Structures in Japan," *Finite Element in Engineering and Science*, Rotterdam: A.A. Balkema, 1997, pp.15-25.
20. Cervenka V: "Reliability-based Non-linear Analysis According to fib Model Code 2010", *Structural Concrete*, Vol. 14, No.1, 2014, pp. 19-28.
21. CEB. "New Developments in Non-linear Analysis Methods," Bulletin No. 229, Lausanne, 1995.
22. Allaix D L, Carbone V I, Mancini G: "Global Safety Format for Non-linear Analysis of Reinforced Concrete Structures," *Structural Concrete*, Vol.14, No.1, 2013, pp. 29-42.
23. Schlune H, Gylltoft K, Plos M: "Safety Formats for Non-linear Analysis of Concrete Structures," *Magazine of Concrete Research*, Vol. 64, No.7, 2012, pp. 563-574.
24. Hendriks M A N: "Nonlinear Finite Element Analysis of Concrete Structures," *JCSS Workshop on Semi-probabilistic FEM calculation*, 2009, Delft, the Netherlands.