ENGINEERS AUSTRALIA

Australasian Structural Engineering Conference

Structural engineering for a sustainable future

30-31 October 2024 | Melbourne | Hybrid

ISBN number for ASEC 2024 is 978-1-925627-76-3

The use of Nonlinear Finite Element Analyses in Structural Design and Assessment

Dr Huy Binh Pham

Technical Principal, SMEC Australia

binh.pham@smec.com

ABSTRACT

Nonlinear Finite Element Analysis (NFEA) has become an indispensable tool in modern structural engineering for the design and assessment of structures. With the increasing complexity of structures and the demand for more efficient and reliable designs, NFEA has been used more and more. This paper provides an overview of the use of NFEA in structural design and assessment, highlighting its capabilities, advantages, and challenges.

The paper begins by discussing the fundamental principles of NFEA, including material modeling, geometric nonlinearity, and contact analysis. It then explores the application of NFEA in research and development where it was first intensively used. The paper also addresses the challenges associated with nonlinear FEA, such as computational complexity, convergence issues, and model validation. It highlights the importance of adopting appropriate modeling techniques, and validation procedures to ensure the accuracy and reliability of the results.

To illustrate these discussions, the paper presents several recent real design and assessment examples in Australia. Through these examples, it offers in-depth analyses and insights gained from practical experiences. Drawing from the author's work over the past two decades, this paper hopes to show the pros and cons of NFEA in contemporary structural engineering practices.

INTRODUCTION

In today's era of increasing complexity of structures and the demand for more efficient and reliable designs, Nonlinear Finite Element Analysis (NFEA) has emerged as a power tool for modern structural engineers. The increasing use of NFEA in the design and assessment of structures, such as bridges and special structures is due to its ability to capture nonlinear behavior, such as large deformations, material yielding, and buckling. However, the importance of incorporating uncertainty and variability into the analysis to ensure robust and reliable structural designs needs to be appreciated.

In this paper, the pros and cons of NFEA are discussed showing its remarkable capabilities as well as highlighting its challenges. Through real case studies from Australia and insights gathered over two decades, this paper hopes to contribute to the efficient and safe use of this method.

NONLINEAR FINITE ELEMENT ANALYSIS

Background

Finite element modelling (FEM) has become an indispensable tool used in structural engineering. FEA for structural applications often involves dividing a structure into ‘elements’ that are connected at 'nodes'. The 'global stiffness matrix', K, is formed by assembling 'element stiffness matrix', k, before solving for displacements. The element stiffness matrix is an intergration of the 'constitutive matrix' E and the 'strain-displacement matrix' B. The integration can be solved numerically using Gauss quadrature, approximated by a sum of weighted values at Gauss points. B is a derivative of the 'shape function', N, which defines displacement at any point within the element from the nodal displacements. Stresses are calculated at Gauss points and interpolate for other locations. It is clear that finite element analysis results are approximated and they can be sensitive to the mesh and the element type or shape function.

In the past, the majority of analyses were restricted to linear elastic. Concrete, however, tends to behave in a non-linear manner primarily due to cracking and other non-linear material behaviours such as concrete crushing at overload and reinforcement bonding slips. Similarly, steel elements tend to buckle and/or yield. Therefore, in many cases, linear elastic analysis is inadequate in describing the complete behaviour of the structure. For the last few decades, there has been much development in the NFEA of reinforced concrete and steel structures and it has now become much more accessible to design engineers. This method has proved to be a powerful tool to study as well as to design concrete and steel structures.

As for FEA, in NFEA, there is a certain level of approximation, which leads to a certain margin of error in the model. The error can be inherent in the method itself as for any numerical simulation. It can also come from the approximations in the mathematical models to represent concrete, steel and bonding between them. In reseach, it is usually necessary to have some test results against which the output from the finite element analysis is validated. In design, it is not always feasiable to have physical test results and other methods of verifications are required.

NFEA needs to be solved by iterative process where the displacement, d, is trialed till the equibrium is reached indicated by meeting the convergence criteria. The following two sections will go into more details in the modelling approaches for concrete and steel structures.

Concrete modelling

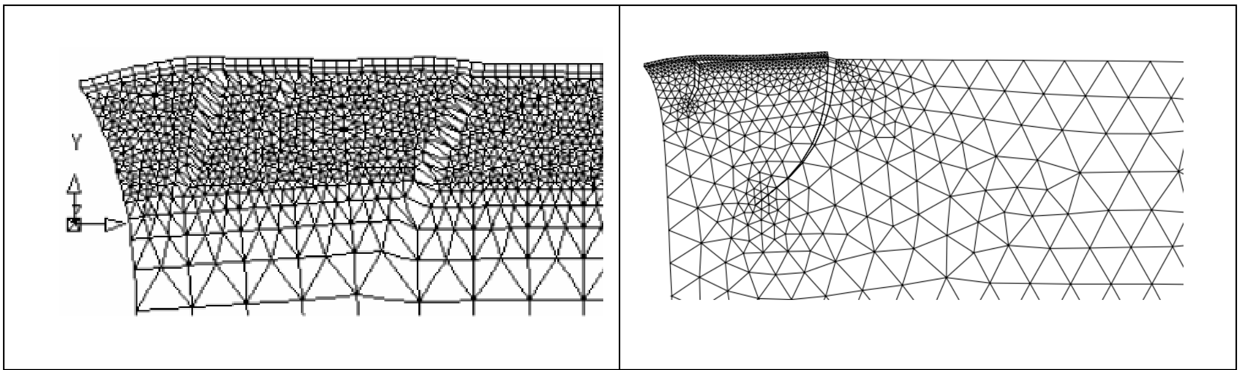
Modelling of concrete behaviour at meso and macro-levels accurately is a challenging task. On the meso-level, normal strength hardened concrete consists of a complex arrangement of inclusions in a more or less homogeneous matrix. The inclusions can be pores and course aggregates. The homogeneous matrix consists of smaller aggregates embedded in a hardened cement paste. Concrete mechanical properties, therefore, can be influenced by a large number of parameters such as the ratio between the matrix and inclusion stiffness, the matrix composition, the type of the aggregates, the texture and roughness of aggregates, the maximum size of the aggregates as well as their size distribution (RILEM Technical Committee 90-FMA, 1989). In a very early age of a concrete structure, i.e. during hardening of the concrete, microcracks may appear due to several sources, such as voids caused by shrinkage or thermal movements and incomplete compaction. When a load is applied, additional microcracks form due to the incompatible deformation of the aggregate and cement matrix. The region around the tip of the cracks is subjected to a high stress concentration and thus a high strain energy concentration. Crack propagation occurs when this energy exceeds the energy capacity of the material. This helps to release any excessive strain energy due to external loading. At or just before the peak load, the existing microcracks start to grow and/or new microcracks originate. With increasing deformation, the microcracks coalesce to form a single macrocrack (Kotsovos and Pavlovic,1995).

On the macro-level, concrete is considered as a homogeneous isotropic material, even though concrete material properties, especially concrete cracking, are linked with the lower structural level, i.e. the meso-level. According to RILEM Technical Committee 90-FMA (1989), there are three basic types of failure: tension (mode-I), shear (mode-II) and tear (mode-III). Since concrete fails most easily in mode-I, cracks tend to branch in the direction of maximum principle stress. This is why mode-I failure has been the focal of concrete cracking research. A mode-I crack propagates when the principle tensile stress exceeds the concrete tensile strength, f_{ct}. Near the tip of a crack, there exists a zone, called 'fracture process zone', over which the stress decays with further displacement. The descending branch is characterised by the specific 'fracture energy', G_f, dissipated during complete crack formation. It is the area under the complete stress-displacement curve (discrete crack) or stress-strain curve (crack band) in pure tension. Consequently, concrete cracking is influenced primarily by two properties: the tensile strength, f_{ct}, and the specific fracture energy, G_f. Both of these properties can vary significantly and are difficult to quantify.

The tensile strength of concrete can be determined by a direct tension test, a cylinder split test or a flexure test. It is more variable than its compressive strength. There are different expressions recommended by different codes to calculate the concrete tensile strength. For example, the predicted tensile strength of the concrete with an average compressive strength of 54 MPa can range from 2.9 MPa to 5.1 MPa.

The fracture energy of concrete, G_f, can also be variable. The fracture energy can be found using testing. There have been a large number of tests from different laboratories, based on different fracture specimens and different testing methods. A wide range of values for G_f has been reported (Bazant and Becq-Giraudon, 2002). G_f has been found to be dependant on several parameters, including but not limited to the aggregate sizes and properties, the water-cement ratio, the test method and the specimen size (RILEM Technical Committee 90-FMA, 1989). To estimate the fracture energy from the basic characteristics of concrete, a number of empirical formulae have been proposed and they produce different values for G_f. For example, the predictions for the concrete with the compressive strength of 54MPa with the maximum aggregate size of 14 mm and the water/cement ratio of 0.59 can range from 0.091 to 0.188 N/mm. There are very limited formulations recommended in the current codes.

In modelling of concrete, compressive and tensile behaviour are often considered separately. For concrete under uniaxial compression, the relation between stress and strain is described by a non-linear curve. One example is the curve by Thorenfeldt et al., 1987. Under biaxial or triaxial compression, concrete shows a pressure-dependent behaviour, i.e. the strength and ductility increase with increasing isotropic stress. The compressive stress-strain relationship for the uniaxial case can be modified to incorporate that behaviour. To model concrete in tension, there are basically two main approaches: smeared crack approach and discrete crack approach (Figure 1). A detailed description of these approaches can be found in RILEM Technical Committee 90-FMA (1989). Briefly, in the first approach, cracks are assumed to be distributed across an element. It is modelled by reducing the stiffness of the element in the direction of the principle strain when the principle tensile strain in an element reaches a limiting value. This approach has been used widely because it enjoys the advantage of permitting automatic crack propagation with a relatively small effort. However, since this method maintains displacement continuity across a crack, a realistic prediction of the distribution of strains in regions adjacent to cracks can be difficult. In the second approach, discrete cracking, displacement discontinuities across a crack are accounted for by disconnecting elements at the nodal points along their boundaries. The main problem in this approach, however, is the difficulty arising from the introduction of additional nodal points required by the altered topology of the analytical model. The cracks usually have to be inserted manually into the mesh.



a) Concrete smear shear crack model in DIANA

b) Concrete discrete crack model in Merlin

Figure 1. Modelling of concrete cracks

Steel modelling

In most cases, steel material is considered as a homogeneous isotropic material. Factors that cause nonlinear behaviour of steel structures are yielding and tension stiffening, buckling and bonding behaviours and they have been extensively studied. Early researches have formulated expressions for design and the nonlinear effects are considered through coeffients such as effective length and moment amplification. Designing steel structures has tranditionally involved simplifying the structures into a collection of individual elements and considerable engineering judgements. With the increase in computer power and availability of sophisticated commercial software, steel structures can be designed or assessed efficiently using NFEA. However, a good understanding of the nonlinear effects is required if this method can be used safely and effectively. These nonlinear effects are described below.

Mild steel has a nonlinear stress-strain response with yielding and strain hardening. The stress-strain curve for high strength steel might not have a distinct yield plateau. At high load levels such as at ultimate limit states, the materials and therefore the structure behave in a nonlinear manner. For a multi-dimensional stress state, the ‘yield criterion’ enables the conversion of a 2D or 3D stress states into an equivalent stress that can be related directly to an uniaxial stress-strain curve. One popular yield criterion is ‘von Mises’.

Another factor that would influence the member behaviour is residual stresses that result from heating or cooling during fabrication. These stresses are self-balancing and, for common section shapes, they have been studied and reported. They can influence the member buckling behaviour and its ultimate capacity. In some cases, the residual stress distribution might need to be investigated and included in the NFEA.

Nonlinear behaviours in steel structures can also be due to large deformations, buckling and imperfection. Large deformations can change the structural shape and its stiffness. This cannot be accurately analysed using linear theory and P-delta analyses are commonly used. This kind of analyses are ready available in many commercial software. This is conventionally allowed for in the standards using moment magnification factors. Buckling can be global such as frame member buckling or local such as plate buckling. The conventional design methods adopt concepts such as effective length factors and effective widths, and some level of simplification is required. Both large deformation and buckling are also influenced by geometric imperfections that are resulted from the fabrication process. Imperfections can amplify the deformation and lead to lower buckling and failure loads. This effect is accounted for some specific cases in the design standard. For other cases, FEA model can be used to quantify the effect where the imperfected shape with the specified amplitude provided in the design codes such as AS5100 can be generated and incoporated in the model. It is important to account for the worst imperfection as some imperfection can even increase the strength.

All of these effects can be modelled using NFEA. However, it is worth noting that the level of modelling complexity is dependant on the design specific need and it is not always necessary to consider all of these effects. For examples, for compact sections, local bucking is not critical and material nonlinearity would be the main focus of the analysis. A good guidance on NFEA for steel structures can be found in DNVGL-RP-C208 (2020).

Solution procedures

In a NFEA, the relation between a force vector and displacement vector is not linear and a solution cannot be found directly. To determine the state of equilibrium, the load is 'applied' incrementally in steps. To achieve equilibrium at the end of each increment, an iterative solution algorithm is used. In the iterative algorithm, the load or displacement is set until an equilibrium is reached meaning convergence criteria are met. The convergence can be based on force, displacement or energy. It is noteworthy that certain procedures or convergence criteria may be more effective for specific problem types compared to others.

Australian Standards requirements

The use of NFEA in design is allowed in Australian Standards. AS5100.5 Clauses 2.3 and 6.6 outline basic requirements for concrete modelling such as the capacity reduction factors, the material mean values and sensitivity checks. Similarly, AS5100.6 Clause 4.10 specifies the requirements for steel modelling. The clause provides details of how to account for geometrical and structural imperfections.

CONCRETE MODELLING EXAMPLES

Reinforced concrete beam restrofited with carbon fibre reinforced polymers (CFRP)

NFEA has been used extensively in research. One example can be found in Pham (2005) where NFEA was used to investigate concrete beams strengthened externally by carbon fibre reinforced polymer composites (CFRP). One model involves the finite element mesh and boundary conditions are shown in Figure 2 below. The mesh, element types and solution procedures were carefully selected as below.

The element size was maintained at approximately 12x12 mm. The element aspect ratios ranged from 1.0 to a 1.2. Since the beam geometry, loading and boundary conditions were symmetrical about the centreline, only half of the beam was modelled. The model was supported vertically at the base and horizontally along the centreline with roller supports. Loading was applied to a single node on the top of the beam. The concrete was modelled using four-node quadrilateral isoparametric plane stress elements (Q8MEM) in DIANA software. Each element has eight degrees of freedom (dof) with two displacements, u_x and u_y, at each node. A 2 x 2 Gaussian integration scheme was used. The steel and CFRP reinforcements in the beams were modelled using beam elements (L7BEN). This element type is a two-node, two-dimensional beam element with the basic variables including the translations u_x and u_y and the rotation φ_z in the nodes. The element was numerically integrated over the cross-section and along their axis. Two-point and three-point Gauss integration schemes were used along the element axis and in the element cross-section, respectively. The reinforcement elements were connected to concrete elements through 2+2 node structural interface elements (L8IF). The normal and shear tractions, and the normal and shear relative displacements across the interface are illustrated in the figure below. Three-point Newton-Cotes integration scheme and linear interpolation were used. The nodes for the steel reinforcement were superimposed on top of the concrete nodes, whereas the nodes for the CFRP reinforcement were offset from the concrete beam soffit by a distance equal to the sum of the adhesive thickness and half of the composite thickness.

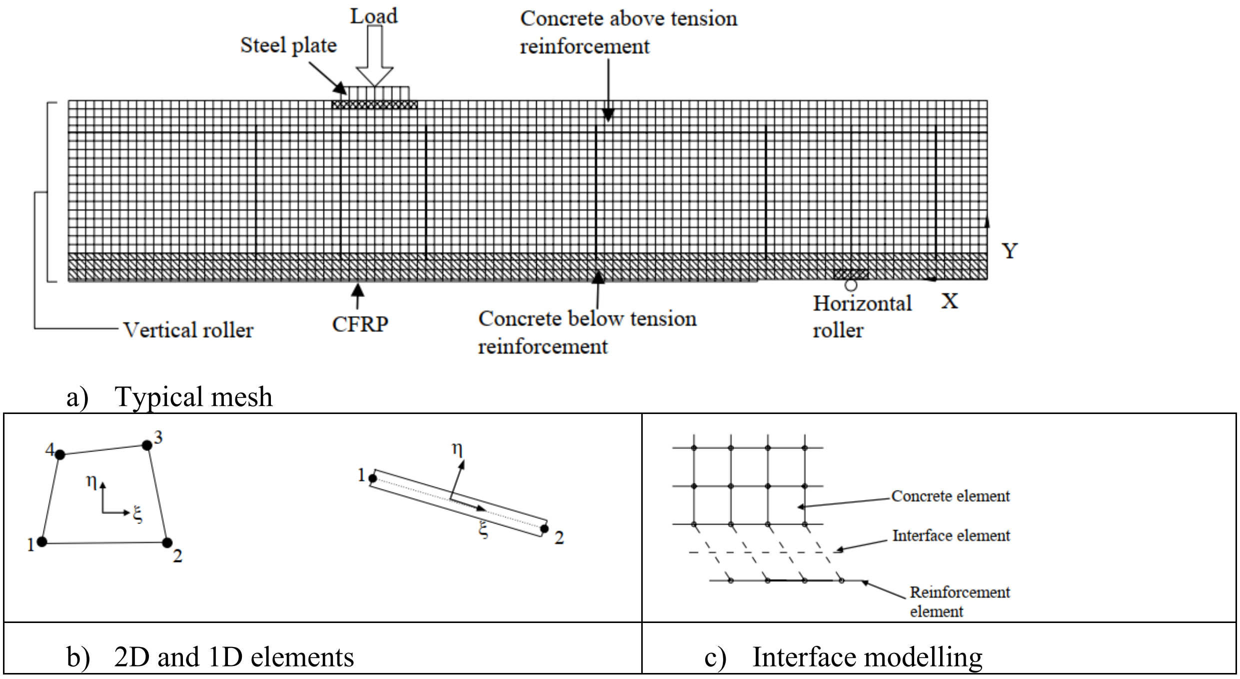


Figure 2. Two-dimensional mesh of FRP retrofitted beams loaded in 4-point bending using DIANA

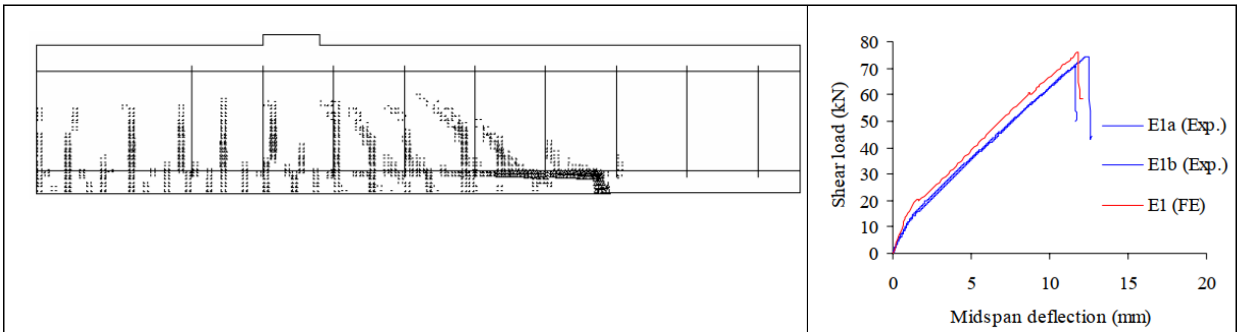
Concrete was modelled using a total strain crack model. The model was developed along the lines of the Modified Compression Field Theory, initially proposed by Vecchio and Collins in 1986. The behaviour of concrete in tension was described using a nonlinear tension softening stress-strain relationship proposed by Hordijk in 1991. The peak tensile strength of the concrete, f_{ct}, was determined using the expression provided by CEB-FIP (1991). The area under the descending branch of the stress-strain curve is given by G_f / h, where G_f is the mode-I fracture energy and h is the crack band width. The crack band width is a discretisation factor, which was determined in DIANA as the square root of the total area of the element. The fracture energy was calculated using the expression by Trunk and Wittman (1998). The behaviour of the concrete in compression was modelled using the function proposed by Thorenfeldt et al. (1987).

The main flexural and shear steel reinforcements in the finite element models were assumed to be an isotropic linear elastic material up until the yield point. Yielding of the reinforcement was based on the yield criterion of Von Mises with strain hardening. In reinforced concrete, the interaction between the reinforcement and the concrete is highly complex. The interaction is governed by secondary transverse and longitudinal cracks in the vicinity of the reinforcement. This behaviour can be modelled with a bondslip mechanism. The bond-slip constitutive law in DIANA is based on a 'total deformation' theory, which expresses the tractions as a function of the total relative displacements. A different bond-slip relation was used to describe the bond between the CFRP and concrete. The bond-slip relation followed a nonlinear curve established previously from shear-lap tests.

In the models, load was 'applied' in steps or increments varied from 0.05 to 0.1mm so that there were about 200 steps before the peak load was reached. At each load step, the Quasi Newton iterative scheme was used to bring the internal forces to an acceptable level of equilibrium. This scheme essentially uses the information of previous solution vectors and out-of-balance force vectors during the increment to achieve a better approximation. This scheme proved to be suitable for the beam models since it converged quickly for most steps. The convergence criterion adopted was based on the energy norm composed of internal forces and relative displacements. A new reference norm was determined at the start of each step. In all of the models, the tolerance for convergence was set to 0.0005 to ensure accurate and good results. The maximum number of iterations for each load step or increment was set to 100. To improve the convergence, the line search algorithm was also used in the study.

To verify that the finite element models simulated the behaviour of the beams properly, four outputs from the experiments and numerical simulations were compared. They were the crack patterns at failure, the load-displacement curves and the strain distributions in the steel and CFRP reinforcements. Good matches were found (Figure 3).

The FEA models were used for other purposes including a detailed parametric study and development of a theoretical models.



a) Concrete crack pattern at failure

b) Comparison of load-deflection curves

Figure 3. Crack pattern and load-displacement curves for RC beams predicted by DIANA

Prestressed concrete box girder

The behaviour of a prestressed concrete box girder bridge was investigated using NFEA. The behaviour was complex especially around the anchorage blocks. To provide a baseline for verification, the bridge was monitored with strain gauges and crack opening gauges, and load tested. The vehicles used for load testing were measured and weighted.

The bridge global behaviours were studied using a three-dimensional grillage model and a shell model. These models are linear elastic and their running time was relatively quick. To study the localised effects around anchorage blocks, submodels were developed using nonlinear brick elements. It was determined that localised non-linearity behaviours do not change the bridge global actions significantly.

Brick models of two locations near the anchorages were implemented in NX FEA software. Several types of modelling were implemented. In the smeared cracking model, all cracks were modelled using smear cracks. In the discrete cracking model, only the cracks in the joints were modelled. In the combined smeared and discrete cracking model, the cracks in the joints were modelled as discrete cracks and the cracks around anchorage block were modelled as smeared cracks. It was found that the joint openings were modelled better using discrete cracks. The predicted crack openings from the discrete and smear crack models were only slightly higher than those predicted by the discrete crack models yet it takes much longer to run (Figure 4). Therefore, discrete and smear crack models were not used for further calibration.

The main findings were:

- Crack opening widths were not uniform across the width of the section.
- Predicted joint openings were relatively close to the measurements
- The predicted crack pattern matched the observed pattern.
- Joint openings were sensitive not only to the applied load but also residual tendon forces and its bonding characteristics.

The study would provide valueable inputs into the design and construction of the strengthening work.

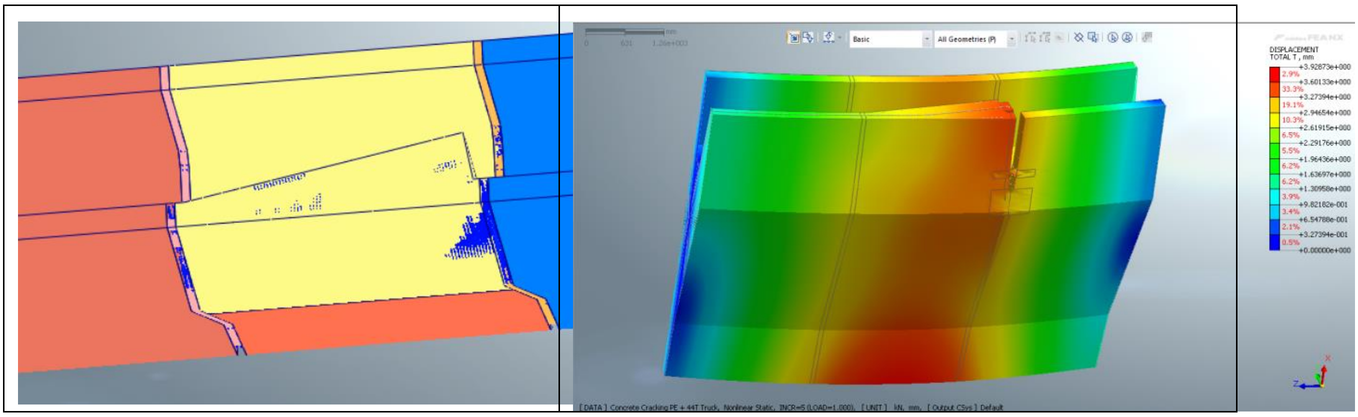


Figure 4. Images of FEA models showing smeared crack patterns and discrete crack opening using Midas NX FEA

Reinforced concrete pile caps

Bridge pile caps are often designed or assessed using strut-and-tie models. In these rather simplistic models, the forces are assumed to follow defined one-dimensional struts or ties. Concrete shear and tensile strength are ignored and the complex three-dimensional effects are often not considered fully.

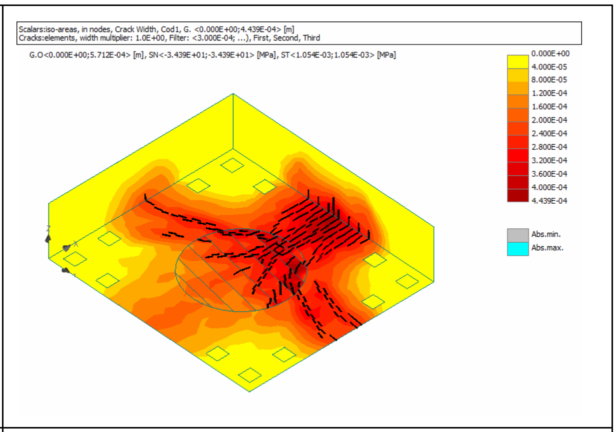
In this example, NFEA was used to assess a series of pile caps to accurately determine if they needed to be strengthened. To overcome the issues with strut and tie modelling, 3D NFEA models were done using ATENA software (Figure 5). ATENA is based on a deformational form of finite element method where the whole structure is divided into finite elements and displacement at any location is approximated by a function of the node displacements.

The material constitutive models in ATENA account for nonlinear behaviour of concrete and steel. The concrete was modelled using CC3DNonlinCementitious2 fracture-plastic constitutive model. In this model, concrete fracture behaviour was modelled using Rankine-Fracturing theory where strains and stresses are converted to crack directions. Crack width was calculated by multiplying the characteristic length L (element dimension) by the fracturing strain. Shear strength of a cracked concrete was calculated using the Modified Compression Field Theory which relates shear strength to concrete compressive strength, crack width and aggregate size. Concrete compressive behaviour was modelled using a plasticity model considering ascending and descending branches.

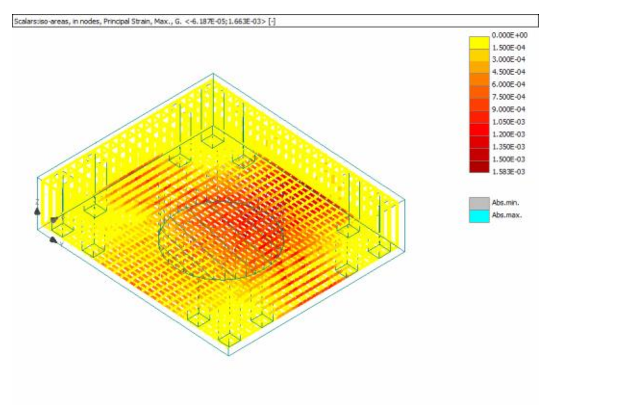
Tetrahedral 3D solid elements were used. Two solvers, and Arc Length, were investigated. The Newton-Raphson method keeps the load increment unchanged and iterates displacements until equilibrium is satisfied while Arc Length iterates increments of both displacement and force. In general the Newton-Raphson method is useful in the sense that the loads and load steps can be defined. However, it can struggle with convergence at high loads when there is significant non-linearity. Arc Length solver typically achieves better convergence but requires much greater memory and time. It was found that both methods to give similar answers and achieve convergence for the parameters of this study. Therefore, Newton-Raphson solver was used more broadly and loads and reactions were checked throughout to ensure equilibrium was achieved. Sensitivity studies on the model behaviour with respect to element size, concrete strength and shear stirrup distribution were carried out.

The NFEA models proved that the pile cap capacities were higher than those predicted by the tranditional strut-and-tie method. This led to significant cost saving for the strengthening work.

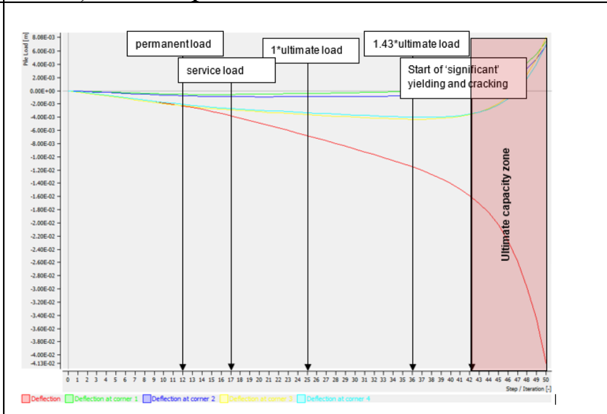
a) 3D pile cap mesh



b) Crack patterns at overload



c) Reinforcement stresses



d) Load vs displacement curve

Figure 5. NFEA models of bridge pile caps using ATENA

External prestressing of super T girders

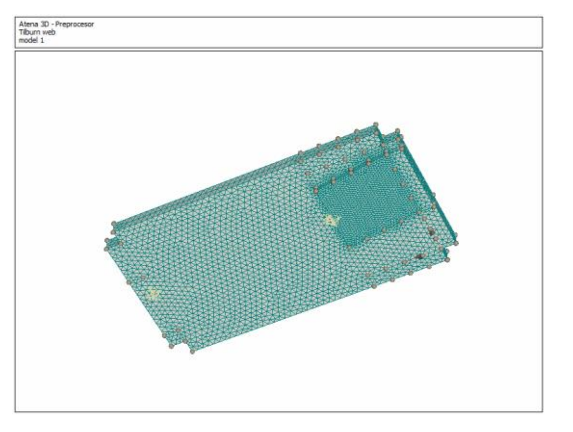
A NFEA model was developed to investigate the causes and the consequences of cracking observed in the web of a super T girder that was strengthened with external prestressing tendons. The web of this girder was subjected to a complex state of stress. Axial forces in the web were due to pretensioned strands, external prestressed tendons (PT) as well as bending of the beam. Vertical shear forces in the web were due to external shear loads. Longitudinal shear forces were also present due to a combination of external shear loads and external PT. The web was also subjected to out of plane bending due to the offset of the PT. It was not possible to analyse this web accurately using conventional linear elastic analysis or strut-and-tie methods.

The steel plate material was represented using the von Mises Plasticity model, and the bond between steel and concrete was modeled using an interface material based on the Mohr-Coulomb criterion with tension cutoff. Reinforcement and its bond were modeled using a bilinear law and the CEB-FIB1990 bond model, respectively.

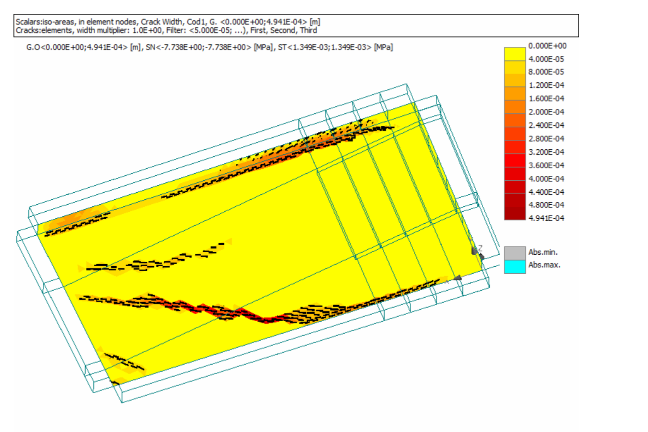
The mesh representing a web segment of 1600 mm long and 800 mm wide, with the boundary conditions reflecting the true conditions as closely as possible. The steel plate, sized 450Wx440D, was included and assumed to be fixed at the outer end. To ensure accurate modeling of out-of-plane bending, a minimum of 2 layers of elements were employed for both the concrete web and the steel plate (Figure 6).

The model verification was also carried out using simpler samples where the predicted ultimate loads were compared with the code values.

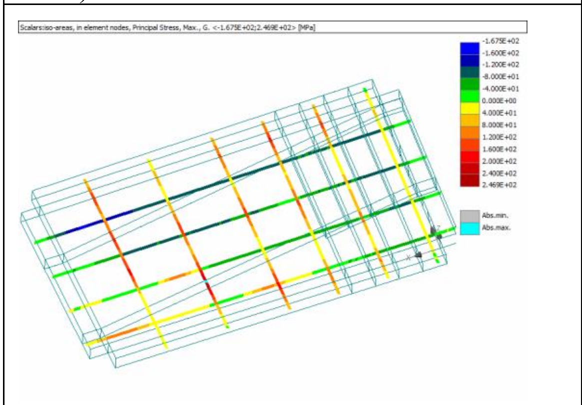
It was found that the model was able to predict both ultimate bending and punching shear loads well. The crack patterns shown by the NFEA matched with the observed pattern. The model proved that once cracks can be grouted, there would be no significant impacts on the beam performance.



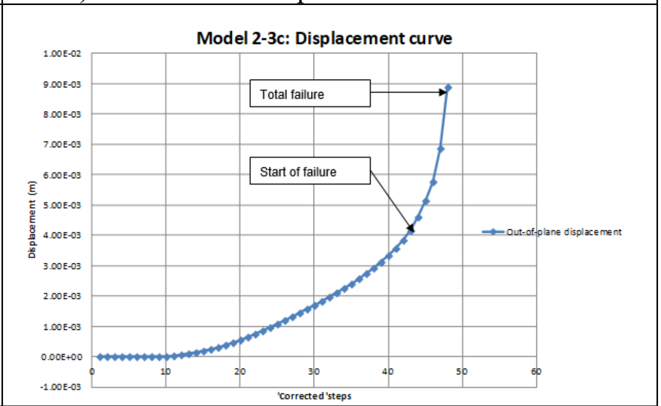
a) Model mesh



b) Concrete crack pattern at ULS



c) Reinforcement stress at ULS



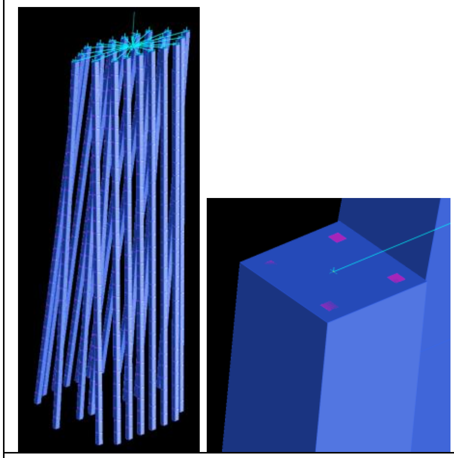
d) Load vs displacement curve

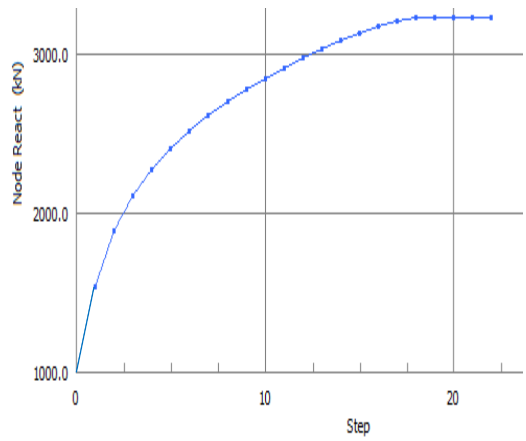
Figure 6. NFEA models of girder web using ATENA

Push-over analyses of pile groups

For bridge design and assessment for earthquake loads, push-over analyses can be used and AS5100 provides guidance on the material models. The analyses are generally straight forwards for monopile foundations as the axial loads are relatively constant. For those cases, the moment-curvature curves can be generated from a section analysis and a nonlinear analyse can be carried out. However, for a pile group subjected signifinant shear loads, the axial loads vary significantly and the moment-curvature method does not work well. This was the case for several large bridges in Melbourne where the pile foundations include a relatively stiff pile cap and a large number of driven precast concrete piles. Hinges were predicted to form in the piles. Push-over analyses were required for a number of load cases which involved significant bending and shear loads.

For those cases, the analyses was carried out using a hybrid section where nonlinear concrete and steel stress-strain curves are include. The composite actions are achieved by ensuring the concrete and steel elements sharing the same nodes. This method proved to work well and the running time was not too excessive (Figure 7). Verification was carried out using the conventional moment-curvature approach for a typical case.





a) Hybrid pile model

b) Load vs displacement curve

Figure 7. Push-over analyses using Strand7

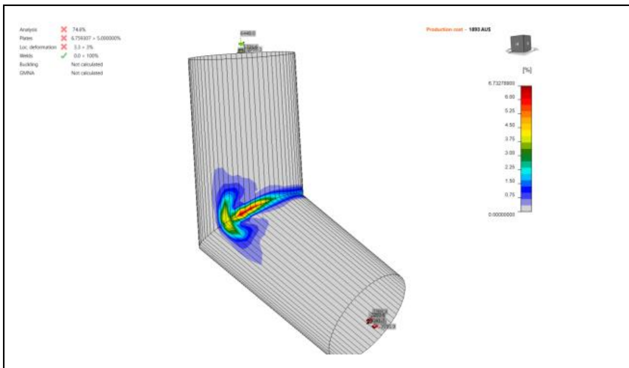
STEEL MODELLING EXAMPLES

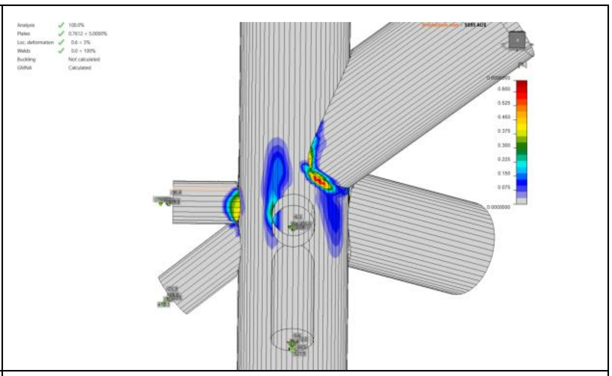
Steel connections

Connections vary greatly and they usually involve bolts, pins, welds as well additional cleats. Bolted connection behaviour is highly nonlinear. The load sharing between bolt is usually not equal and end bolts tend to take more loads. Welded connections is more rigid and the weld stress distribution is rarely uniform. Tranditional analytical methods involve some level of simplification.

The design of connections has been made easier with the increase availability of specialised FEA software. Connections can be modelled fairly quickly using software such as IDEA Statica (Figure 8) or Strand7. The geometric complexity can be modelled and yielding of materials can be considered. Slippage can also be modelled and tolerances are usually ignored.

In this example, various connection of a large steel gantry were modelled using NFEA. They included welded connections between tubular segments, welded connection between gusset plates and tubular section, bolted spliced connections, base plate connections… Most models were done in IDEA Statica and then verified using independent Strand7 models. Good matches were found and the NFEA was able to pick up design issues such as nonlinear buckling and load redistribution.





a) Plastic strain contours in a knee joint

b) Plastic strain contours at a node with multiple tubes

Figure 8. Modelling of steel tube connection using IDEA Statica

Steel to concrete composite connection

Several tall steel pole connections were to be strengthened using a complex composite connection involving steel jacket, grout and concrete. This connection relies on multiple factors including friction, bearing, grout and concrete shear and tensile strength. It is not possible to account for these factors using hand calculation and NFEA was utilised.

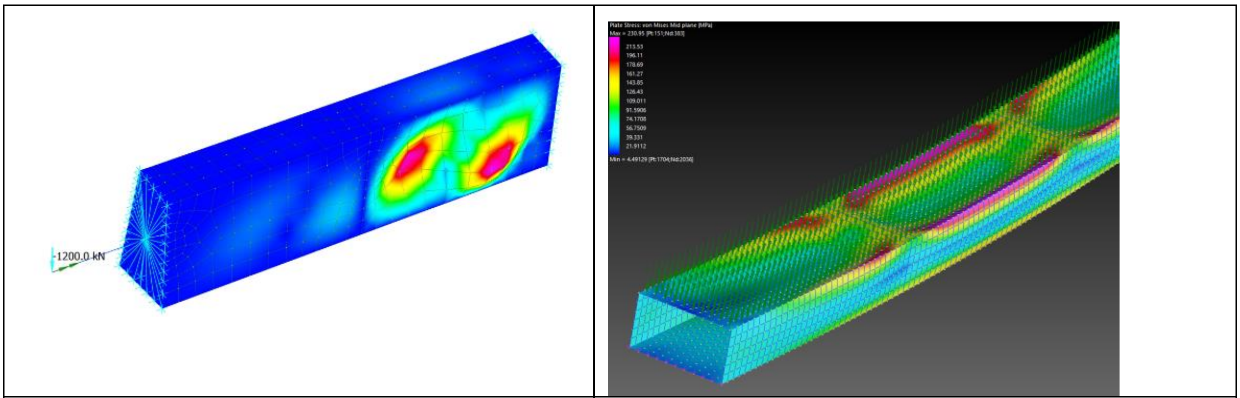
The FE models used for design review purposes were created in FEA NX software package. Relatively fine mesh was used to capture the thin layer of grout as well as bearing rings. Steel was modelled using 'Non-linear Von-Mises' material model whereas concrete and grout were modelled using 'Non-linear concrete smeared crack' model. The interfaces were modelled using 'Coulomb friction-concrete' model.

The NFEA models appeared to be able to capture the expected nonlinearities. However this is an example where NFEA needs to be used with great caution given the complexity and the unproven construction technique.

Stiffened box girders

Girders with transverse and longitudinal stiffeners can be designed in accordance with AS5100.6. The method adopted in the standard is based on checking isolated stiffened steel panels. These panels' behaviour is typically highly nonlinear and often involves buckling with significant post-buckling strength. The method stated in the standard applies for a number of common designs under typical loading condition. Simplications are inherent in the method. For complex sections, the girder performance can be determined using NFEA where the geometrical and material nonlinearities can be considered. Stiffeners and diaphragms can also be checked using the NFEA model. Imperfections and residual stressses should be considered. Verification using standard formulas should also be carried out.

NFEAs were used in various design and assesment work for box girders (Figure 9). These analyses were often verified by comparing with available emperitical design charts. However, they could be done for virtually any box and stiffener shapes and arrangments. The graphical outputs often provide great insight into the potential failure modes and 'weak' areas that require more attention. Analyses for imperfections and residual stresses can be complex but it is worth noting that the capacity of stiffened box girders is not very sensitive to these two effects and they can be considered in the model uncertainty factor.



a) A FEA model to study torsional behaviour of complex section

b) NFEA model of a rectangular box with uniform top loading

Figure 9. Stiffened box girder checks using NFEA Strand7 models

SOIL STRUCTURE INTERACTIONS

Buried culverts

In this example, several buried culverts were required to be assessed. The culverts were supporting multiple lanes of road traffics. To accurately assess the culvert performance, it was modelled and analysed in the finite element analysis software NX FEA. The culvert was modelled in both 2D and 3D. Sensitivity analyses were also undertaken for the models using a range of general material properties as well as support conditions. The ground was modelled using Mohr-Columb model. The interface between the outside surfaces and the fill was modelled using interface elements. The culvert and its foundation were modelled as linear elastic elements. All construction stages were considered. The results were verified by comparing with independent Plaxis models.

The models were able to simulate three-dimension effects from complex geometry as well as the complex loading pattern. It was also able to predict the interaction between the structure and the ground. This method was proven to be a much more effective way to design and assess buried structures as compared to the conventional 2D elastic modelling method.

Land bridges

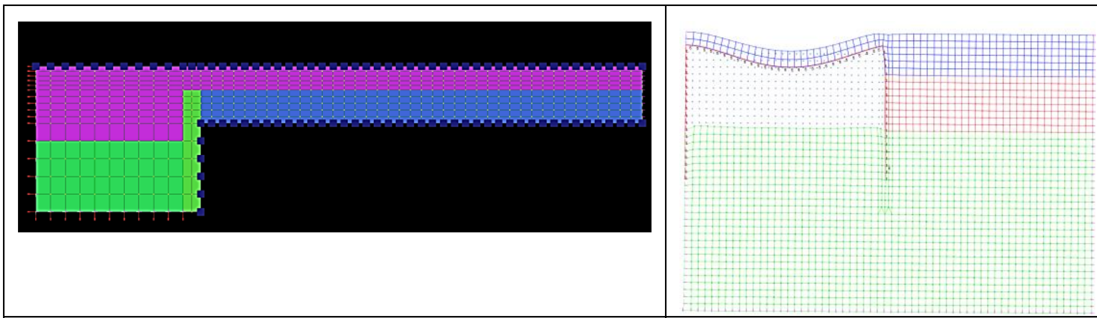
Land bridges are bridges supporting a thick layer of fill for plantation. Designing integral land bridges presents distinct challenges, including temperature effects, creep and shrinkage, soil-structure interaction, and ground settlement, for which there are limited studies and design guides available. NFEA models were used in this case study of a bridge crossing a freeway.

The bridge is approximately 45m wide and accommodates four traffic lanes as well as a footpath. The bridge comprises two 35m spans and utilizes a superstructure made up of 2m deep super T beams, along with a 250mm thick cast in-situ deck. The bridge is integrally connected to both the abutment piles and pier piles to eliminate the need for maintaining and replacing bearings and expansion joints, which would be challenging given the fill depth. The bridge is situated in Melbourne, and the ground profile is comprised of a 4m thick layer of soil over weathered rock. The fill layer typically measures 1.5m deep, and no approach slabs are proposed. The bridge FEA models using Strand7 software were used to look at thermal loads as well as soil-structure interaction (Figure 10).

Thermal loads were applied both at the top and along the bridge soffit, and were based on BOM data for Melbourne. The bridge's initial temperature was assumed to be 20°C. It was determined that the critical scenario for the bridge's average temperature occurs during heatwaves, which can cause the bridge temperature to rise and affect its performance. According to the analysis results, the bridge's average temperature rose by 6.63°C, which is significantly lower than the 50°C limit specified in AS5100. The top bridge temperature, which was used for calculating the temperature gradient, reached 37°C, which is also similar to the value predicted by a international standard (BS EN 1991-1-5). The soffit temperature was higher than what the standards specified, but this can be attributed to the aforementioned assumption. Similar outcomes were observed for the scenario of a typical Melbourne cold snap.

To study the soil-structure interaction and ground movements during and after construction, a 2D finite element model was built. In the model, several construction stages were included. The ground was modelled using 2D Plane Strain Elements with Mohr-Coulomb Soil Constitutive Model. The abutment walls, bridge deck and ground anchors were modelled using Beam Elements. The contact between the wall and the ground was modelled using Contact Elements. As the bridge is symmetrical, only half to the bridge was modelled and the boundary condition was set appropriately.

This model was used to calibrate a global model which was used to design member sizes and connections. The model also showed the anticipated ground settlement at the back of the abutments. For the case study, it was found that the settlement was small enough and an approach slab was not required.



a) FEA thermal model

b) FEA soil-structure interaction model

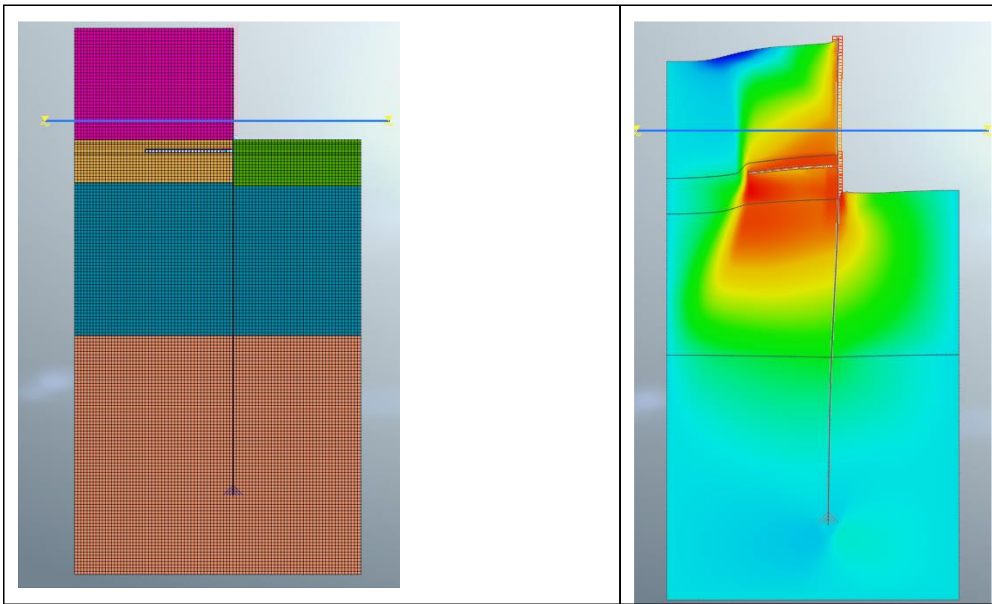
Figure 10. An example of FEA models for land bridges

Retaining walls

While retaining walls can be typically designed using hand calculation or spreadsheets, in many cases, significant soil-structure interaction can occur and NFEA is required. One example is for a tall retaining wall with a relieving slab.

It is common to model retaining walls using Plaxis software. However, in this example, NX FEA software was used. The ground was modelled using 2D Plane Strain Elements with Mohr-Coulomb Soil Constitutive Model. Construction stages were considered.

The model was able to simulate the significant soil-structure interaction and the construction stagin effects (Figure 11). The design of the structural elements was carried using the actions from this NFEA model.



a) FEA mesh for a retaining wall

b) Deflected shape predicted by NFEA

Figure 11. An example of FEA models for retaining walls

SUMMARY

NFEA is a powerful engineering tool that tackles complex structural behaviors. It allows modeling of phenomena like concrete cracking, steel yielding, construction stages, buckling, soil-structure interaction, and various geometries. As shown in this paper, NFEA can be applied to analyse bridges, retaining walls, buried structures, and retrofitting projects. However, for efficient and safe use, an understanding of both NFEA and core engineering principles is crucial. Careful model checking, verification, and sensitivity studies are essential.

The growing adoption of NFEA holds promise for improved design efficiency and safety in the structural engineering field.

REFERENCES

Bazant, Z. P. and Becq-Giraudon, E., *Statistical prediction of fracture parameters of concrete and implications for choice of testing standard*, Cement and Concrete Research, Vol. 32, No. 4, pp. 529-556, 2002

Comite Euro-International du Beton, CEB-FIB Model Code 1990. London, Great Britain, Thomas Telford, 1991

DNVGL-RP-C208 "Determination of structural capacity by non-linear finite element methods" DNV GL AS, September 2019

Kotsovos, M. D. and Pavlovic, M. N., *Structural concrete - Finite element analysis for limit-state design*. New York, Thomas Telfold, 1995

Trahair, E. and Bradford M. A., *The behaviour and design of steel structures to AS 4100*, E & FN SPON, London and New York, 1998

Thorenfeldt, E., Tomaszewicz, A. and Jensen, J. J. "Mechanical properties of highstrength concrete and applications in design". Proc. Symp. Utilization of High-Strength Concrete (Stavanger, Norway), Tapir, 1987

Trunk, B. and Wittmann, F. H. "Experimental investigation into the size dependence of fracture mechanics parameters". Third international conference of fracture mechanics of concrete structures, D-Freiburg: Aedificatio Publ., pp. 1937-1948, 1998

RILEM Technical Committee 90-FMA, *Fracture mechanics of concrete structures - From theory to applications*. London, Chapman and Hall Ltd, 1989

Pham H. B. *Debonding failure in concrete members retrofitted with carbon fibre reinforced polymer composites* PhD Thesis, Monash University, 2005

Pham H. B. "Design of integral land green bridges" 11th Australian Small Bridges Conference, 2023

Standards Australia (2017) AS 5100.5:2017: Bridge Design, Part 5: Concrete

Standards Australia (2017) AS5100.7:2017: Bridge Design, Part 6: Steel

Strand7 Program Manuals and Webnotes, 2024

Zhang, Z. J., Chen, B. S, Bai, R. and Liu, Y. P., "Nonlinear behaviour and design of steel structures: review and outlook", *Buildings 2023*, 12, 2111

BIOGRAPHY

Dr. Huy Binh Pham is a Technical Principal at SMEC Australia. He has 25 years of experience in structural design and design management across projects in Australia, the Middle East, and Southeast Asia. Dr Pham has research and teaching skills. He is a specialist in strengthening structures using fibre-reinforced polymers and in advanced modelling of concrete and steel structures. He has contributed to over 30 publications, served as a guest lecturer at universities, and is a recognised reviewer for international journals.