University of Surrey

Faculty of Engineering and Physical Sciences

Civil and Environmental Engineering

Long-Term performance of a T-shaped girder bridges by using 1D, 2D and 3D FE modelling approaches - a comparative analysis

Emanuele Canestro

6454825

Supervised by: Dr Helder Sousa

August 2018

A dissertation submitted in partial fulfilment of the requirements of the Degree of Master of Science in Civil Engineering

'I confirm that the dissertation entitled ‘A comparative analysis on the Long-Term performance of a T-shaped girder by using 1D, 2D and 3D FE approaches’ for the partial fulfilment of the degree of MSc in Civil Engineering has been composed by myself and has not been presented or accepted in any previous application for a degree. The work, of which this is a record, has been carried out by myself unless otherwise stated and where the work is mine, it reflects personal views and values. All quotations have been distinguished by quotation marks and all sources of information have been acknowledged by means of references including those of the internet.

Student’s name: Emanuele Canestro

Date: 21st August 2018

Acknowledgments

I would like to thank my project supervisor, Dr. H.Sousa, for his effort, kindness and guidance and for this wonderful possibility.

I would like to thank Prof P. Bocchini and Prof A. Strauss, for their external support by sharing documentation and data related to the case study herein explored, as well as for the opportunity in participating in this joint-collaboration project that supported my dissertation.

Abstract

In today's world, there is a huge pressure towards sustainable transportation networks. As part of a critical type of assets within these transportation networks, prestressed concrete large-span bridges play a vital role, which are nowadays expected to perform up to 100 years, at least. Despite the fact that the main deterioration processes are related to fatigue originated by the traffic loading and the materials ageing due to corrosion phenomena, the long-term performance of this type of bridges has becoming a up-to-date topic of research due to the collapse of the Koror-Babeldaob Bridge (in 1996). The reasons to its collapse is stills under discussion among the experts on the field of prestressed concrete structures. The time-dependent properties of concrete and prestressing steel, the time-history related to the numerous construction phases, and the applied loads are found to be critical parameters. Nevertheless, it has also become evident that an accurate investigation on how to better model these structures, based on Finite Element Analysis (FEA) method, is mandatory. Measurements from the field have been showing that 1D modelling is not perhaps the most reliable approach to assess the long-term performance of prestressed concrete bridges.

In this context, and further to a scaled T-shaped girder beam that has been tested and observed, in laboratory conditions, is herein analysed by using different FEA approaches with the aim to discuss the differences that it is indeed committed when analysing the long-term performance of prestressed concrete bridges. More precisely, three types of FEA models have been considered, mainly: (i) 1D model with beam elements, (ii) 2D model with plane stress elements and (iii) 3D model with brick elements. A throughout discussion is then promoted concerning the long-term behaviour. Also, the time invested in developing an accurate FEA model was assessed in order to better decide which approach is more suitable in a cost-benefit basis.

The results herein presented clearly shows that the pattern of the long-term vertical displacements is highly dependent on the FEA approach used, with the 1D beam model underestimate significantly these deflections. It is also clear that the development of advanced 3D FEA models is mainly mandatory for bridges with a huge socio-economic impact.

UNIVERSITY OF SURREY

INDEX

List of FIGURES

LIST OF FIGURES

1. Introduction

1.1 An overview of the subject

In today's world, there is a huge pressure towards sustainable transportation networks. As part of a critical type of assets within these transportation networks, prestressed concrete large-span bridges play a vital role, which are nowadays expected to perform up to 100 years, at least. Despite the fact that the main deterioration processes are related to fatigue originated by the traffic loading and the materials ageing due to corrosion phenomena, the long-term performance of this type of bridges has becoming a up-to-date topic of research due to the collapse of the Koror-Babeldaob Bridge (in 1996).

1.2 Literature gap

The reasons to its collapse is stills under discussion among the experts on the field of prestressed concrete structures. The time-dependent properties of concrete and prestressing steel, the time-history related to the numerous construction phases, and the applied loads are found to be critical parameters. Nevertheless, it has also become evident that an accurate investigation on how to better model these structures, based on Finite Element Analysis (FEA) method, is mandatory. Measurements from the field have been showing that 1D modelling is not perhaps the most reliable approach to assess the long-term performance of prestressed concrete bridges.

This research follows previous studies which, through laboratories' tests, tried to understand the behaviour of the long-term performance of prestressed concrete bridges. Many factors and phenomena have been identified to have an influence on the long-prediction behaviour, as aforementioned.

In this context, and further to a scaled T-shaped girder beam that has been tested and observed, in laboratory conditions [4], is herein analysed by using different FEA approaches with the aim to discuss the differences that it is indeed committed when analysing the long-term performance of prestressed concrete bridges. More precisely, three types of FEA models have been considered, mainly: (i) 1D model with beam elements, (ii) 2D model with plane stress elements and (iii) 3D model with brick elements. A throughout discussion is then promoted concerning the long-term behaviour. Also, the time invested in developing an accurate FEA model was assessed in order to better decide which approach is more suitable in a cost-benefit basis.

1.3 Dissertation structure

With the focus on the quantification of the long-term performance of prestressed concrete bridges, by means of controlling displacements and stresses in critical section, this dissertation is structured as follows

Firstly, a literature review is presented with the objective in contextualizing the reader to the problematic under discussion, as well as, highlighting the importance of these phenomena along the bridges' lifetime and therefore, how this work might contribute to a better knowledge on this with benefits for the maintenance of these structures along their lifetime (i.e. 100 years).

Secondly, a description of the methods used to support this analysis is done, mainly by focussing on the modelling approach at three levels: (i) material, (ii) FEA and (iii) loading. Highlights are mainly given to the approach for the FEA modelling mainly by considering three different approaches, mainly (i) 1D models based on Timoshenko beam elements, (ii) 2D models based on plane stress elements and (iii) 3D models based on brick elements. A brief reference to the software used is done, DIANA [11], which is perhaps the most advanced and well recognized software package to perform long-term analysis of prestressed concrete structures.

Thirdly, the application case is introduced – a T-shape girder beam that has been tested at University of Natural Resources and Life Sciences, Austria [4]. The option for this, and not for a full-scale bridge, is mainly related to the fact that the complexity of such problem is so high, that at this stage of knowledge, simpler scale models would be beneficial to better understand what is behind this unexpected behaviour of this type of structures. In addition, this reduced model has been monitored over time, which is a plus in order to further compare the obtained results in this work.

Finally, the results obtained from the three different FEA models are compared and discussed further. Mainly on the basis of displacements, tendon forces and stresses in concrete. The main conclusions and further steps close this piece of work

Also, it is worthwhile to mention that this work tries to emphasize an important aspect, that perhaps is not so highlighted, which is related to time investment required to develop these type of models, i.e. a discussion towards a cost-benefit basis. In other words, the time that was necessary to spend by the author was recorded along the development of this work. Hence, this relevant information can then be also included in a rational and efficient discussion on how to better model this so important and crucial assets on the transportation networks.

2. Literature review

In today's world, the current progress towards sustainable transportation demands for a prolonged life-time, sometimes more than 100 years, for concrete large-span bridges. In order to accomplish this goal, in the last decades the development of many studies of the long-term analysis broke out in the engineering area of interest. Concrete is a composed material which goes through over the time in terms of amplification of the stress distribution and deformation. These aspects are the most important phenomena that current engineers are trying to handle in understanding the concrete behaviour. Measurements, obtained since early stages with specific sensors would be an important tool to get a better comprehensible knowledge of the real behaviour of the structures. Many studies have shown that the long-terms predictions, given by any codes, differ notably from the observed structural response due to the evaluation of the shrinkage and creep during the design phase and the variability concerning temperature and humidity changes. Therefore, data from an accurate monitoring system is an important tool that allows the engineers to validate the design assumptions, calibrate the models and to update the safety coefficients of the codes.

The optical fiber sensors [1]-[2], following the development of this technology, have increased dramatically their presence and use in the engineering field. The small size, the ease of installation, the high performance in terms of measurements and precision and the possibility to be integrated in wide and complex network systems are important advantages which provide benefits surpassing the high cost of using this advanced tool. The main function of the optic fiber is the ability of making measurements depending on the variations of external parameters which induce a variation of the light guided by the fiber. These parameters might be displacements in the concrete, cracks propagation, strain, temperature changes etc. There are many types and solutions of optic fiber. Fiber Bragg grating sensors are the most commonly used system in civil structures.

In order to provide a physical interpretation of the FBG, transducer systems are developed at the same time. The main objective of a transducer is to transform the attribute representative of the global or local behavior of the structure into an optical nature signal such as, the wavelength related to the FBG.

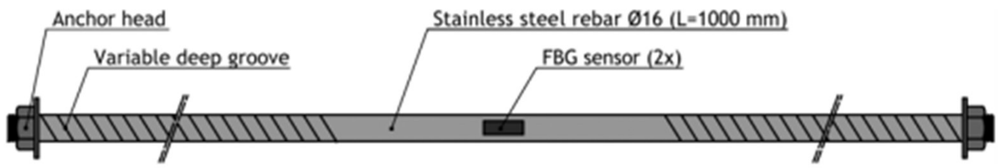


Figure 1 - Optical strain transducer to be embedded in the concrete

The installation of these systems which have to be embedded in the cast is complex and usually it begins prior specific laboratory tests which allow calibrating and assessing these instruments. Once ready, the complex network system can start working providing quasi-instantaneous (depending on the interval of the measurements) output results which can be constantly seen and monitored by users through PC. Therefore, it permits to monitor the concrete behavior due to the actions of the traffic (live loads) and intervene when a value of the strain or deformation overcome the safety limits gaining time and save money.

2.1 Concrete modeling

As we have seen, the sensors placed in the cast allow the researchers and the users to monitor constantly the behavior of the structure. Consequently, in the last years it was tried to do a step further, namely understanding and studying the long-term performance of the concrete. In the prestressed there are four main phenomena which usually occur during its life: Creep, shrinkage, cracking and steel relaxation.

Creep (sometimes called cold flow) is the tendency of a solid material to move slowly or deform permanently under the influence of mechanical stresses. Generally, there are two types of creep, mainly (i) static creep and (ii) cyclic creep.

Static creep is the normal viscoelastic behaviour of the concrete while the cyclic creep is produced by live loads, but it is often neglected in the previsions (this part is not discussed in this paper).

Shrinkage is the volumetric changes due to the loss of moisture content and it may induce forming cracks in the material.

The codes have always tried to evaluate these two phenomena. The following formulas show how the Eurocode 2 assess creep, shrinkage and the compressive strength of the concrete considering the parameters functions of the time:

- Compressive strength: $f_{cm}(t) = \beta_{cc}(t) f_{cm};\ \beta_{cc}(t) = \exp[s\left(1 - \sqrt{\frac{28}{t}}\right)]$ (2)
- Shrinkage: $\varepsilon_{cs}(t) = \varepsilon_{cd}(t) + \varepsilon_{ca}(t)$ (3)
- Creep: $\varepsilon_{cc}(t, t_0) = \varphi(t, t_0)\, \varepsilon_c(t_0);\ \ \varphi(t, t_0) = k_{cc,0}\varphi_0[\beta_c(t, t_0)]^{k_{cc,t}}$ (4)

Therefore, in the long-term performance creep and shrinkage tend to change over the time. Especially in bridge constructed [3] by stages the analysis of the creep is based on the assumptions that the stress varies in step like shape and is constant during construction stages but only changes at initial time of each construction stages. Therefore, the creep effect is calculated taking into consideration each increment (at every stage change) of both the creep and the shrinkage. Hence, the Sum of the initial stress and incremental stress is used to calculate the creep over each time interval. These effects may also induce additional compresses forces in the cast increasing prestressing losses and if the tensile stress exceeds the strength of the concrete it may generate cracks and expose the reinforcement bars to an aggressive environment.

Consequently, a health monitoring system is not only useful to provide instantaneous values but also helps to assess the behaviour of these mentioned phenomena which are commonly present during the life-cycle of the prestressed concrete with the aim of improving the performance. As results of that, multiple research [4]–[6] tried to evaluate and make predictions regarding the changes happening in the cast through the use of Finite Element packages which are important tools to help, validate and control the obtained results, predictions or assumptions that have been made.

2.2 Steel modelling

In the prestressed structures, the concrete, as mentioned above, is not only the material which is influenced by the time but even also the steel bars and tendons embedded inside it. Steel relaxation [7] is a phenomenon that usually occurs on steel tendons, stretched between two fixed points, which lose a part of their initial tension due to creep propagation. The presence of the creep is commonly taking place during the first days and usually it stops, under a constant strain, after about 15 days. There are many methods to mitigate the creep propagation in the steel such as using galvanized wires or overstressing the steel about 10% above its initial stress and progressively releasing it to the initial stress. There are also several methods, provided by codes, which try to assess the relationship between the rheological properties of the concrete and the relaxation of the steel such as the lump-sum individual methods (ASSHTO) and the time step methods. The first one tends to estimate the losses of the creep, shrinkage and steel adding them up in order to get the total loss. On the other hand, time-step method divides the time into small intervals and the prestressed loss in each interval is added up to get the total loss. This method is more accurate but more complicated and time consuming compared to the first one. Unfortunately, current solutions and methodologies tend to consider these three phenomena which have an influence on the concrete strength losses, separately, without providing a proper tool which considers the real interactions between creep, shrinkage and steel relaxation.

2.3 Geometric modelling

Currently, as noted above, with the development of the technology, new modern structural Finite elements software broke out in the professional field helping the engineers to validate and asses their calculation and assumptions. Especially, the software packages are able to make many type of analysis and investigate multiple aspects related to this dissertation project such as the stress and deformation changes over the time, for example, of the previously specified in the introduction, T-shaped concrete beam. Usually, the analysis can be split into three main levels of accuracy (Fig.2) related to the geometry [8], namely beam elements 1D, 2D shell elements and 3D solid elements mesh, depending on the engineering needs or design requirements.

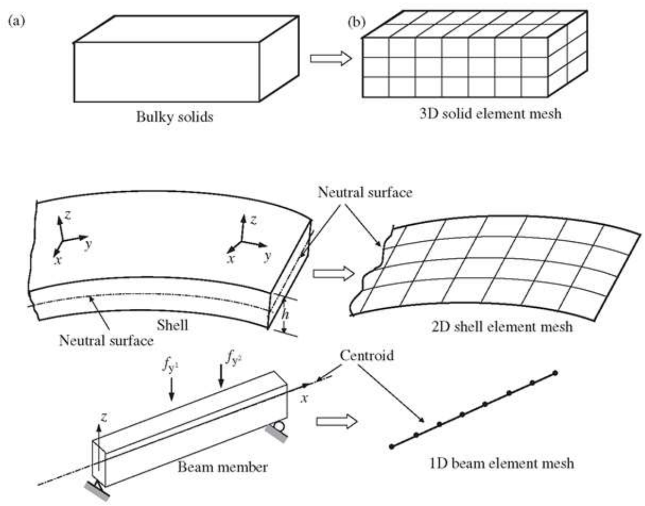


Figure 2 - 1D beam element, 2D shell and 3D solid mesh

Beam elements, is the simplest model which can be used. Generally, it has three degree of freedom and simply provides information regarding the bending moments, shear, axial forces and deformation occurring in the elements. It is generally used to model simple beams, columns, frames etc.

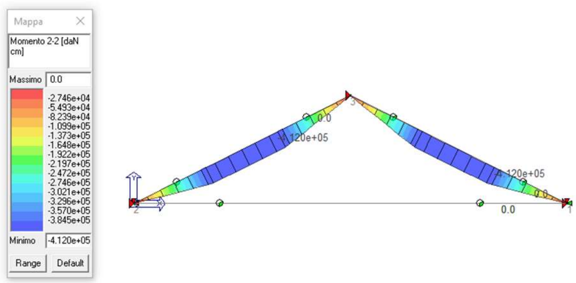


Figure 3 - 1D beam element, truss structure bending moment

On the other hand, shell elements (5-6 DOF) and 3D solid mesh make a deeper and more accurate analysis about the localization of the stress (i.e. located in the flange or in the web), local deformations due to a certain type of load, cracks propagation, temperature effects etc. Consequently, these models are generally built up to simulate the behaviour of single elements such as pipes, plates, beams etc. Usually, shell elements are used when two dimensions are much bigger of the third one, (i.e. steel pipes where the thickness can be neglected, plates etc.), hence the stress is located only on the surface of the mesh. While, when the three dimensions are comparable is preferable using a solid mesh (i.e. Concrete pipes, where the thickness should have to be taken into account), hence the propagation of the stress is spread across the entire element.

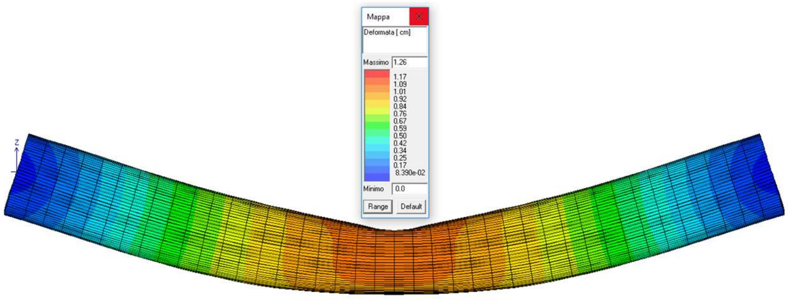


Figure 4 - 2D Shell element, steel pipe deformation

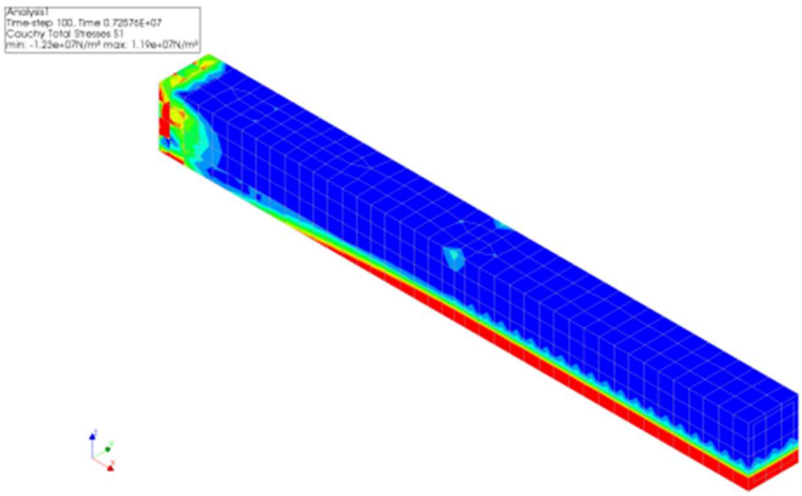


Figure 5 - 3D solid mesh, Concrete beam, stress propagation

2.3. FEA software packages suitable for long-term analysis

Today, there are many software packages commercially available which can perform a long-term analysis:

2.3.1. Sap2000

Sap2000 [9] is probably the most well-known software in the academic environment. Since the first version, introduced almost 40 years ago, the SAP name is synonymous with state-of-the-art analytical methods. Today's SAP2000 continues the tradition of the time. SAP2000 is produced by Computer and Structures inc. of Berkeley, California. Thanks to its highly qualified staff of professional engineers, researchers and distinguished representatives of the academic world, CSi has been at the forefront of technological development in engineering for four decades. With CSi products you can be sure to use the finest software available, supported by a company with an incredible history of innovation and dedication to the complex needs of the profession.

SAP2000 is a finite element calculation program designed primarily for civil engineering. It has very versatile characteristics, so much so that it falls into the category of so-called "general" programs, i.e. capable of analyzing structures with very different characteristics, such as dams, communication towers, sports stadiums, industrial plants, buildings and much more.

Advanced Analysis Techniques allow Non-Linear Step Analysis for Large Deformations, Multiple P-delta, Eigen and Ritz Analysis, Cables, Only Tense or Compressed Elements, Instability Analysis, Fast Non-linear Analysis (FNA) with Dampers, Base and Plastic Support, Control of Movements with the Virtual Work method, Construction by Phases and much more.

2.3.2. Midas FEA

Midas FEA [10] is a robust 3D modeling software and advanced nonlinear civil engineering analysis. The software has the latest modeling tools and, thanks to its compatibility with other formats, it is possible to achieve very advanced modeling levels in a short time. The powerful mesh maker and solver allow accurately representing and solving the finite element model that best represents the previously created solid geometry.

Midas FEA allows you to create very complex geometric objects thanks to the presence of the latest generation 3D geometric modeling tools.

Advanced non-linear solutions of different types of structures: historical masonry, reinforced concrete, steel instability through the use of different solvers: static linear / phase-based / non-linear for material and geometric, dynamic analysis / heat transfer / heat of hydration / fatigue analysis.

2.3.3. DIANA FEA

The FEA software, used in this paper, is DIANA [11]. DIANA (acronym Displacement Analyzer) is a Finite component Analysis (FEA) problem solver developed and distributed by DIANA FEA BV (previously TNO DIANA BV) and several other resellers worldwide. The computer code is used at each ends of the market, by tiny consultancies and world engineering consultants, analysis establishments and is used by several extremely revered academic establishments worldwide in each civil and geotechnical engineering courses. DIANA is provided with powerful solvers that allow the analysis of a large vary of structures, massive and small with basic or advanced analyses. An oversized choice of material models, component libraries and analysis procedures are obtainable inside the package which provides DIANA an oversized degree of flexibility. Especially, the most fields of use of DIANA include style and analysis of dams and dikes; tunnels and underground structures; oil, gas and historical constructions and huge ferroconcrete structures. A number of the specialized analyses obtainable in DIANA for these fields of use include unstable analysis; fire analysis and young hardening concrete.

Consequently, this software has been chosen for this dissertation. Its simplicity, according to the following mentioned case study [6], of mashing, positioning the reinforcements and tendons, considering the different phases of analysis and levels of accuracy and its intuitive interface allow to have an important tool to be used and very defined results to analyze in order to make important comparisons with laboratory test measurements and, in some cases, other finite element operator systems.

2.4 Cases found in the literature

Many researchers and scientists have been trying in the last decades to analyze the concept previously described, related to certain type of structural work in site or a single prestressed beam, through FEA software using several models differing from geometry, depending on the analysis purpose with the aim extrapolating right information to assess the long-term performance of the cast.

Table 1 – Main up-to-date references found in the literature

Year	Key contents	Reference
2018	Steel relaxation long-term behaviour in the prestressed concrete	[7] T. Guo, Z. Chen, S. Lu, and R. Yao, "Monitoring and analysis of long-term prestress losses in post-tensioned concrete beams," *Measurement*, vol. 122, pp. 573–581, Jul. 2018.
2018	Combined shear and long-term behaviour of a T-shaped beam modelled with a FEA software	[4] A. Strauss, B. Krug, O. Slowik, and D. Novak, "Combined shear and flexure performance of prestressing concrete T-shaped beams: Experiment and deterministic modeling," *Struct. Concr.*, vol. 19, no. 1, pp. 16–35, 2018.
2016	Sap2000 manual	[9] CSI, "SAP2000. Analysis Reference Manual," *CSI: Berkeley (CA, USA): Computers and Structures INC*. 2016.
2016	Long-term performance of concrete bridge considering cyclic creep	[5] T. Tong, Z. Liu, J. Zhang, and Q. Yu, "Long-term performance of prestressed concrete bridges under the intertwined effects of concrete damage, static creep and traffic-induced cyclic creep," *Eng. Struct.*, vol. 127, pp. 510–524, Nov. 2016.
2016	Solid and shell elements theory	[8] T. Sussman, "Solid and shell elements."2016.
2013	Assessment and long-term prediction based on monitoring data	[6] H. Sousa, J. Bento, and J. Figueiras, "Construction assessment and long-term prediction of prestressed concrete bridges based on monitoring data," *Eng. Struct.*, 2013.
2011	Long-term analysis of rheological parameters	[3]Q.-J. Wen, "Long-term effect analysis of prestressed concrete box-girder bridge widening," *Constr. Build. Mater.*, vol. 25, no. 4, pp. 1580–1586, Apr. 2011.
2011	FBG sensor embedded in the cast. Health monitoring systems	[1]B. Torres, I. Payá-Zaforteza, P. A. Calderón, and J. M. Adam, "Analysis of the strain transfer in a new FBG sensor for Structural Health Monitoring," *Eng. Struct.*, vol. 33, no. 2, pp. 539–548, 2011.
2010	Long-term concrete behaviour based on FBG sensors measurements	[2] C. Rodrigues, C. Félix, A. Lage, and J. Figueiras, "Development of a long-term monitoring system based on FBG sensors applied to concrete bridges," *Eng. Struct.*, vol. 32, no. 8, pp. 1993–2002, Aug. 2010.

Therefore, it has been decided to show and describe highlighting the main features and key contents of reference [4]-[5]-[6] because they are based on monitoring data assessing the long term-performance of the prestressed concrete on site of real structures or single beam element useful for the main scope of this dissertation.

2.4.1. The couple effect of the static and cyclic creep

The coupled-effect of the static and cyclic creep, according to reference [5] , is often neglected in previsions. Therefore, in the paper it was tried to model a Bridge using FE software (ABAQUS) using 3D solids elements in order to make a Rate-type formulation, based on rheological models. The creep can be generally expressed by the function j(t,t'), where t' is the age when the stress is exerted and t is the current time. However, the function is not a convolution type. Therefore, all the history variables have to be stored at the actual time step. This is very an expensive prediction of the concrete losses in massive structures. Consequently, this goal can be accomplished through rheological models (Kelvin chain Fig.6) in order to convert the integral-type formulation into a rate-type one, in which the previous history can be fully described by the internal variables of the last time steps.

The following formula is giving a general idea of the Kelvin chain, as expressed by Eq. (6):

$$\frac{1}{E''} = \frac{1}{E_0} + \sum_{\mu=1}^{N} D_{\mu}^{-1} = \sum_{\mu=1}^{N} A_{\mu}(1 - \lambda_{\mu}) \qquad (6)$$

Where D_{μ}^{-1} are distinctive modulus of multiple Kelvin units, E'' is the incremental modulus of concrete at a current time-step, E_0 is the instantaneous modulus, N is the total number of Kelvin units, λ_{μ} are related to the retardation time τ, A_{μ} are the discrete spectra obtained according to the given creep function. The great achievement of that equation is that from the values of E'' it is possible to build a 3D stiffness matrix enabling the use of 3D solid elements in the software for box girders with the aim of minimizing the accuracy of the 1 or 2D simplification.

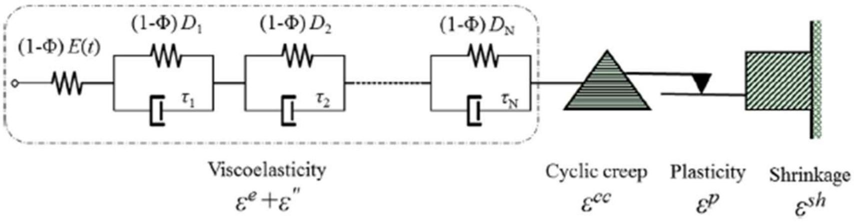


Figure 6 - Kelvin chain

The bridge under this study is the Humen Bridge in Guandong Province in China (Fig.7). It is a 3 span prestresssed concrete bridge erected by segments showing a main span length of 270 m. It also carries all type of vehicles, especially heavy tracks making the structure an important and notable example of the coupled effect of the creep.

The model (Fig.8) consists in two types of simulation in order to compare the results to real measurements previously given by sensors. The first model (Fig.9 left) doesn't take into account the coupled effect considering just the static creep while the second one (Fig.9 right) tries to understand the real differences considering even the cyclic creep.

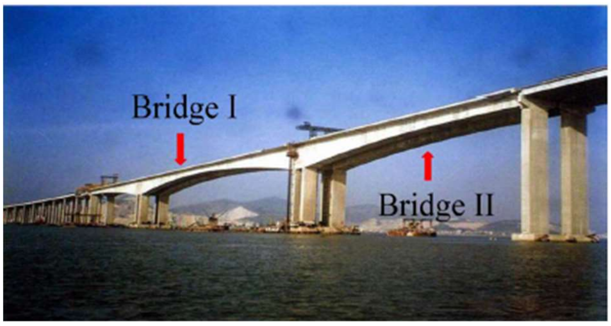


Figura 7 – Bridge view

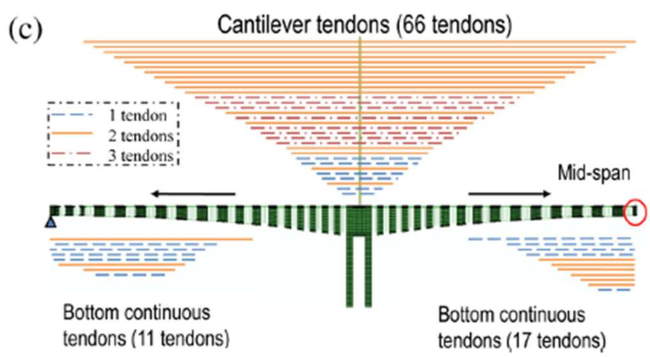


Figure 8 – Tendons and bridge model

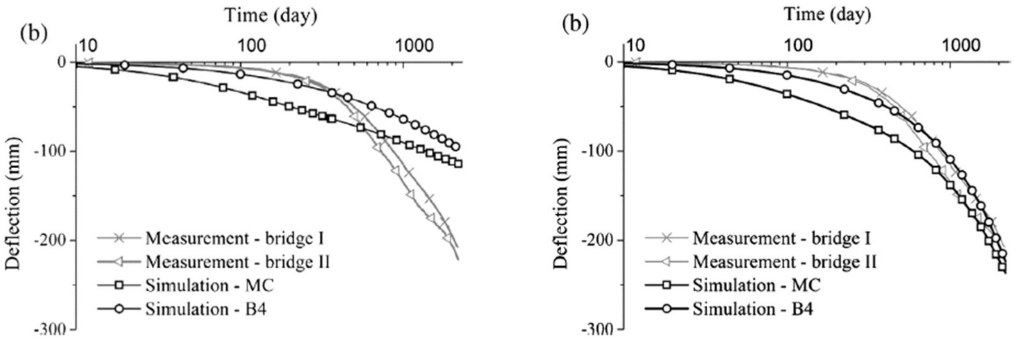


Figure 9 – Results and Comparisons between real measurements

The results were plotted in two Log-scale graphs where they show a comparison between real measurements and the two different creep models. The right graph gives information about the coupled effect of the creep. Clearly, the simulation is very close to the real sensors data. On the other hand, according to the left graph, considering just the static creep occurring in the concrete tends to underestimate the real deflection (mm) of the material over the time.

2.4.2. Assessment and long-term prediction of prestressed bridge

This study [6] tries to understand and have a better view of the differences between the national codes based on monitoring data collected on the field. Over the years, with the development of the technology, the monitoring system became more accurate, offering information regarding multiple structural aspects like durability, integrity and reliability providing an optimal maintenance planning and safe operations. Hence, the measured data has become essential in order to validate the design assumptions, to calibrate the structural models and to update the safety factors in the codes. The case study is based on the analysis of a box girder bridge named Lezìria Bridge (Fig.10), Portugal. It was constructed in 21 months and its main span is approximately 970 m. This structure is equipped by a sophisticated system of sensors placed into two different positions (Fig.11). A non linear analysis has been carried out by using structural software (DIANA). Only one alignment was modelled in two dimensional analyses because of the alignments of each slab and girders are similar. In this model, the geometry and loading are symmetrical to the principal girders' axis. The steel bars in the concrete (reinforcement) were modelled using embedded reinforcements elements in order to take into account the prestressed losses due to creep and shrinkage (Fig.12). Especially, the tendons' modulus of elastic is characterized by considering the average value of the prestressed cables used for each span. Generally, the value considered into numerical model is 199.7 GPa which satisfies the EC2 [12] limits (195-210 GPa). The piles of the bridge were modelled considering an elastic interaction between the soil and the structure. As results of following the chronology of the construction, a phased analysis was performed with 56 stages. The mechanical parameters of the concrete were based on the tests of 150 mm cubes. The expressions of the evolution of the concrete compressive strength, creep and shrinkage are given by EU2 as following:

- Compressive strength: $f_{cm}(t) = \beta_{cc}(t) f_{cm}$; $\beta_{cc}(t) = \exp[s\left(1 - \sqrt{\frac{28}{t}}\right)]$ (7)
- Shrinkage: $\varepsilon_{cs}(t) = \varepsilon_{cd}(t) + \varepsilon_{ca}(t)$ (8)
- Creep: $\varepsilon_{cc}(t, t_0) = \varphi(t, t_0)\, \varepsilon_c(t_0)$; $\varphi(t, t_0) = k_{cc,0}\varphi_0[\beta_c(t, t_0)]^{k_{cc,t}}$ (9)

Where t represents the concrete age in days, s is a cement-hardening coefficient, $f_{cm}$is the mean value of the concrete compressive strength at the age of 28 days, $\varepsilon_{cs}(t)$ is the total shrinkage strain divided in two parts, namely, $\varepsilon_{cd}(t)$ and $\varepsilon_{ca}(t)$ respectively the drying shrinkage and the outogenous shrinkage strain. φ_0 is the notional creep coefficient and $\beta_c(t, t_0)$ is a coefficient which describes the variation of creep over the time after the loading.

Figure 10 - Leziria Bridge under construction

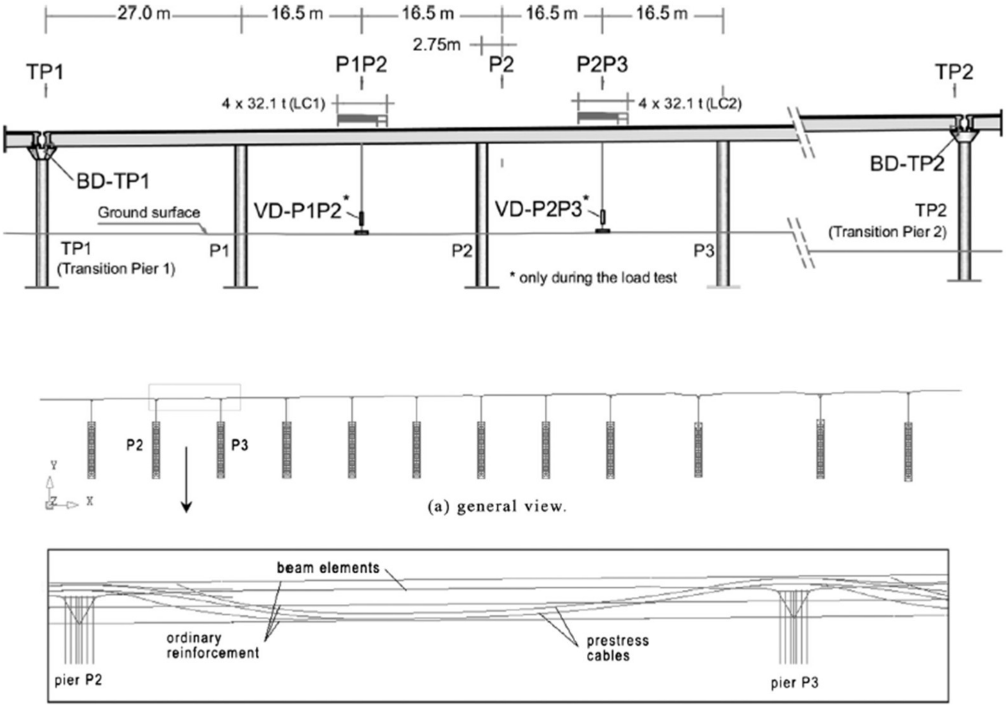


Figure 11 - Bridge model. Reinforcements and tendons layout

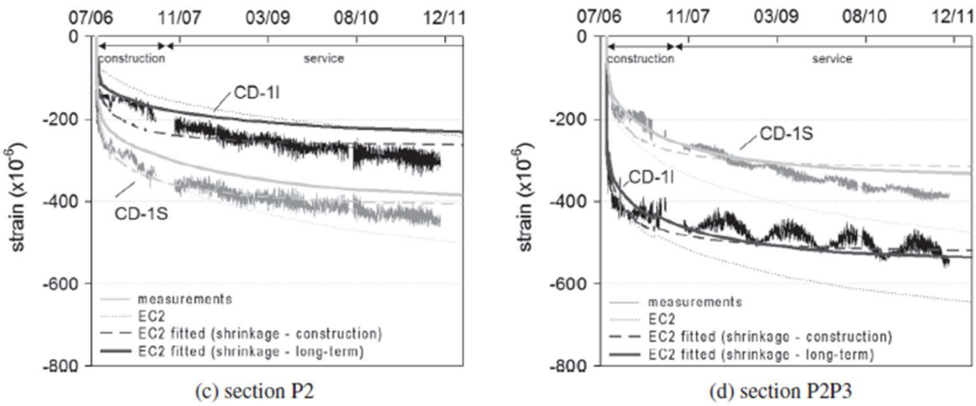


Figure 12 - Results and comparisons

Clearly, according to Fig.12, the long-term analysis results provide evidence that Eurocode2 tends to overestimate the strain behaviour over the time. This is, certainly, favourable for the safety but on the other hand it may increase the costs of the Bridge due to oversized cross-sections and elements. The fitted model is much closer to the real measurements even if there are still some slight differences. One of the reasons could be due to the sensors near the top face of the deck girder which are exposed to particular conditions because of the bituminous layers tend to increase the temperature in the concrete and reduce the humidity. Obviously, it was noticed the higher is the sensitivity of the sensors to the variations of the temperatures, better is the adapting with the results of the numerical model. Consequently, it can be argued that the importance of comparing real measurements to models, based on generic national annex or codes, is essential in order to understand the real behaviour of concrete with the aim of implementing and revising these coefficients and parameters which usually are taken into account at the design stage.

2.4.3. Combined shear and flexure performance of prestressing structures

This paper will discuss in the next chapters about modelling a determined prestressed T-shaped beam. The beam was previously analyzed by Dott.Alfred Strauss in the reference [4]. It was tried to analyze and understand the combined shear and flexure response of the beam with the aim of focusing on the fracture-mechanical parameters. Usually, they are considered as the most critical and important concrete parameters for realistic modelling. Particularly, tensile strength and fracture energy are essential in non linear finite element modelling due to their role in crack initiation and propagation. In this research, like the previous mentioned studies, many fracture tests were carried out in laboratories on 10 scaled beam 5 m long. Some sensors were installed on selected longitudinal reinforcement bars and on stirrups embedded in the concrete matrix of the shear field area in order to get real measurements and indentify the mechanical parameters. Normally, these deterministic parameters have a sort of lack over the time. Therefore, stochastic models were taking into account considering every single mechanical and fracture parameter as a variable. In total, 134 concrete samples were tested at different times of 1, 7, 28, and 126 days in order to determine the stochastic values and properties (E_c, f_{ct}, G_f, and f_c) as shown in Fig.13.

The FEA model was carried out by the software Athena3D. The shape of the cross-section is slightly different compared to the real one, namely using a polygonal transition between the web and the top flange of the beam (Fig.14). The reinforcements and tendons (Fig.15) were modelled considering one-dimensional reinforcement material defined by the stress-strain diagram below with a comparison with the real properties of tendons (Fig.14).

Fracture test—standard storage
C50/60
14 specimens: 7·$\beta_{BZ\ 100\times100\times400}$ (f_c, E_c, G_f); 7·$G_{F\ 150\times150\times150}$ (G_f)
tested: 28 days (β_{BZ}, G_F)
C40/50
similar to C50/60[1]
Specimens cast along with laboratory beams C50/60
19 specimens: 10·$\beta_{D\ 150\times150\times150}$ (f_c); 5·$G_{F\ 150\times150\times150}$ (G_f); 4·$\beta_{BZ\ 120\times120\times360}$($f_c$, E_c, G_f) tested: 7 days (β_D), 21 days (β_D, G_F, β_{BZ})
Fracture test—component storage
C50/60
70 specimens: 21·$\beta_{D\ 150\times150\times150}$ (f_c); 28·$\beta_{BZ\ 100\times100\times400}$($f_c$, E_c, G_f); 21·$G_{F\ 150\times150\times150}$ (G_f)
tested: 28 days (β_{BZ}, G_F); 7 d (β_D, β_{BZ}); 28 d (β_D, β_{BZ}, G_f); 126 days (β_D, β_{BZ}, G_f)
C40/50
similar to C50/60[1]
Specimens cast along with laboratory beams C50/60
120 specimens: 6·$\beta_{D\ 150\times150\times150}$ (f_c); 3·$\beta_{EM\ 100\times100\times360}$ (E_c); 3·$\beta_{SZ\ 120\times120\times360}$ ($f_{ct,sp}$)
tested: 28 days (β_D), 33 days (β_D, β_{EM}, β_{BZ})

Figure 13 – casting and testing procedure for the determination of the stochastic properties

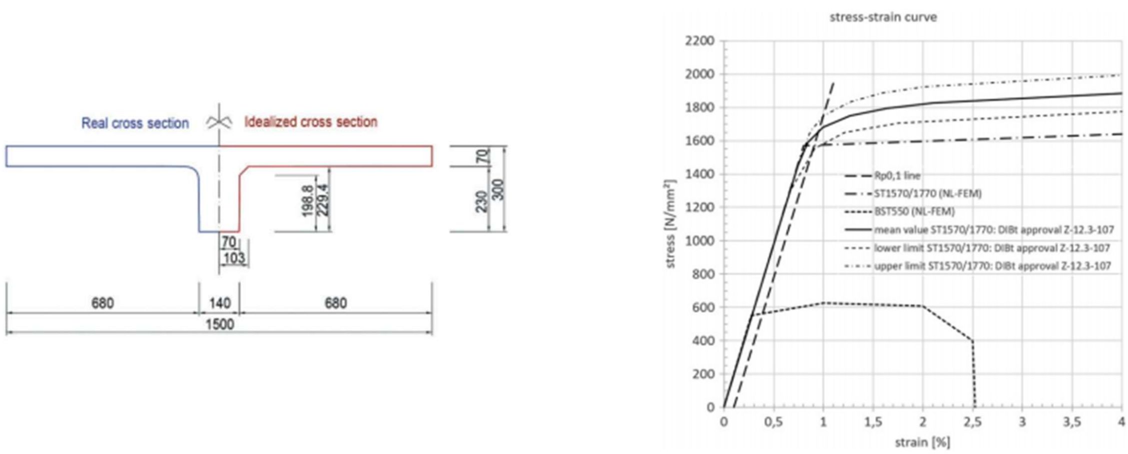


Figure 14 – Real and idealized cross section. Stress-strain diagram.

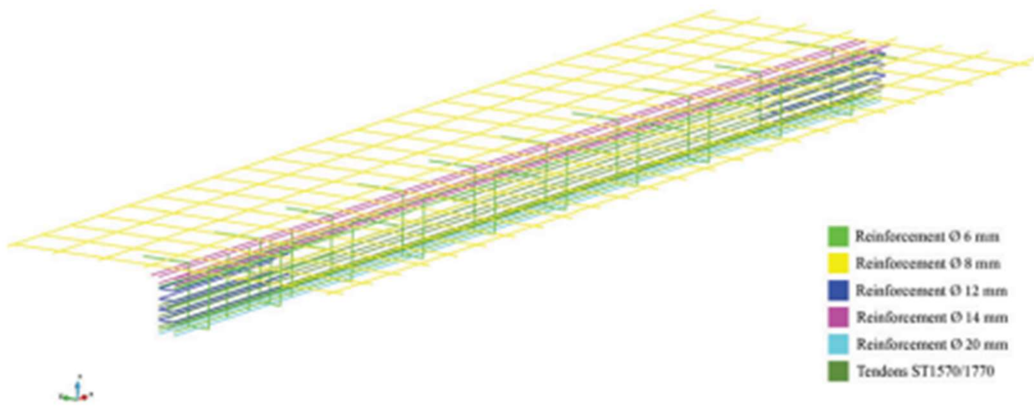


Figure 15 – Tendons and reinforcements

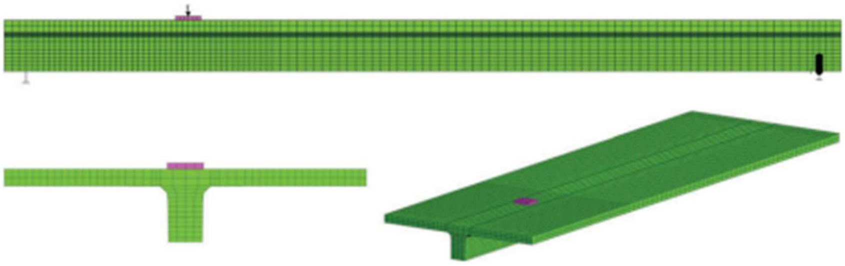


Figure 16 – Generated mesh, supports and load plate

The loads application was spread into four intervals. The 1st interval involves the application of live loads. The second one defined the prestressing forces which were applied immediately after 14 hours during the manufacturing process. Therefore, the concrete young's modulus was roughly the half compared to the one after 28 days of hardening and the initial strain of the reinforcement's line was reduced by the difference between the concrete strain at 14 h and 29 days. The third interval is the application of the temperature loading with the aim of simulating the rheological response of the precast concrete element. Obviously, creep and shrinkage deformation play a main role about the material strength losses. The last interval is the loading force. There are two ways of loading, namely, using forces or displacements. The second one allows obtaining the post peak behaviour while force-loaded model doesn't. Many models of the T-shaped beam were made considering a different level of details from V1 to V7. Fig.17 shows the differences with the tests results.

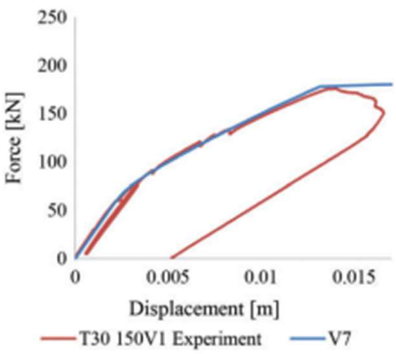


Figure 17 – Comparisons between model V7 and experiment

Clearly, the graph provides information about how the rheological parameters allow having a good approximation of the concrete behavior before the post peak plastic field.

2.3. The motivation of the project

This project (see Fig.18) takes the cue from the previously mentioned studies. Especially, the same T-shaped beam of reference [4] is analyzed and similarly modeled using a different structural software package. This research is trying to do a step forward. The analysis is carried out by using different detailed levels of accuracy for the chosen models. Especially, the software allows building up the beam using two dimensional analyses, shell elements and even generating a 3D mesh composed by brick elements. Obviously, modeling through finite elements takes time and unfortunately, nowadays, there are just few studies which are facing this aspect. Consequently, this paper tries to investigate this matter with aim of comparing the results given by a different accuracy, possibly compatible with future real measurements provided by a laboratory test. This highlights the fact that it is very useful estimating the percentage of accuracy for each models and understanding the invested time into the software. Particularly, it is assumed that modeling with 3D elements provides a theoretical percentage of 100% as accuracy. Therefore, one of the main scopes is to analyze the magnitude of stresses and deformations generating in the concrete and understand if it is worth or not spending a certain amount of time on modeling with a different level of detail. The following scheme (Fig.18) gives a hint of the project that will be described in the next chapters.

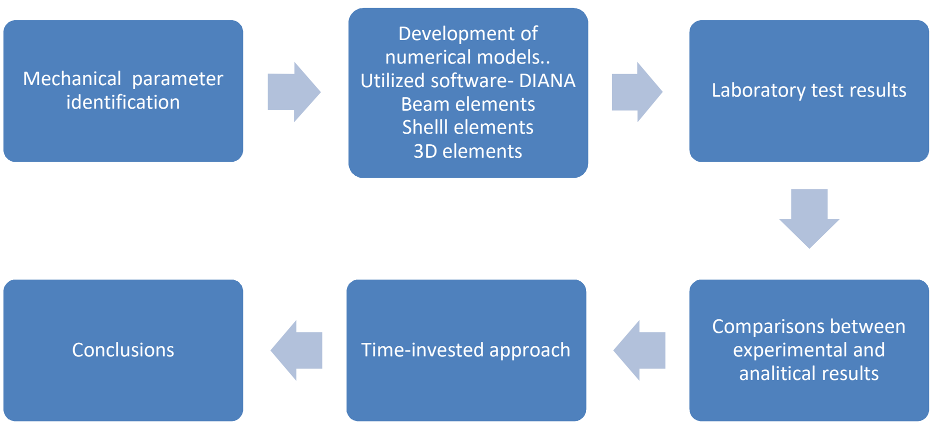


Figure 18 – Brief scheme outline of the project.

3. Description and methodology

For the scope of this project, a phased non-linear finite element analysis has been carried out by using the finite element code DIANA [11]. The software has a wide database regarding the concrete modeling such as the Eurocode 2 EN 1992-1-1, the American Concrete Institute 209R-92 model code, the American Association of State Highway and Transportation Officials (AASHTO) and the fib Model Code for Concrete Structures 2010[13]. For the analysis the last model Code has been chosen. The expressions proposed by the fib Model Code for Concrete Structures 2010 are consider for this project. Mainly, the time-dependent properties of concrete are briefly described.

3.1. Concrete modelling

3.1.1. Evolution of concrete compressive strength

When subjected to sustained high compressive stresses the compressive strength of concrete decreases with time under load due to the formation of micro-cracks. This strength reduction is counteracted by a strength increase due to continued hydration. The combined effect of sustained stresses and of continued hydration is given by Eq.9 and Eq.10.

$$f_{cm,sus}(t,t_0) = f_{cm}.\beta_{cc}(t).\beta_{c,sus}(t,t_0) \tag{9}$$

$$\beta_{c,sus}(t,t_0) = 0.96 - 0.12\{ln[72(t-t_0)]\}^{\frac{1}{4}} \tag{10}$$

Where, $f_{cm,sus}(t,t_0)$ is the mean compressive strength, $\beta_{cc}(t)$ is the development function, $\beta_{c,sus}(t,t_0)$ is a coefficient which depends on the time under high sustained loads t-t0 in days. t_0 is the age of concrete at loading and $(t-t_0)$ is the time under high sustained loads in days.

3.1.2. Modulus of elasticity

The variation of the Young's modulus over the time can be estimated as following:

$$E_{ci}(t) = \beta_E(t). E_{ci} \tag{11}$$

$$\beta_E(t) = [\beta_{cc}(t)]^{0.5} \tag{12}$$

Where $E_{ci}(t)$ is the modulus of elasticity in MPa at an age t. E_{ci} is the modulus at an age of 28 days and $\beta_E(t)$ and $\beta_{cc}(t)$ are coefficient which depend on the concrete age.

3.1.3. Shrinkage

The initial and creep strain components are defined consistently, so that their sum results in the correct load-dependent strain. Therefore, the evaluation of the creep phenomena can be estimated as following:

$$\varepsilon_{ci}(t_0) = \frac{\sigma_c(t_0)}{E_{ci}(t_0)} \tag{13}$$

$$\varepsilon_c(t) = \varepsilon_{ci}(t_0) + \varepsilon_{cc}(t) + \varepsilon_{cs}(t) + \varepsilon_{cT}(t) \tag{14}$$

Where $\varepsilon_{ci}(t_0)$ is the initial stress, $\varepsilon_{cc}(t)$ is the creep strain at time $t>t_0$, $\varepsilon_{cs}(t)$ is the shrinkage strain and $\varepsilon_{cT}(t)$ is the thermal strain.

3.1.4. Creep

The creep is assumed linear related to the stress within the range of $|\sigma_c| < 0.4 f_{cm}(t_o)$

$$\varepsilon_{cc}(t_0) = \frac{\sigma_c(t_0)}{E_{ci}(t_0)} \varphi(t, t_0) \tag{15}$$

$$\varphi(t, t_0) = \varphi_{bc}(t, t_0) + \varphi_{dc}(t, t_0) \tag{16}$$

Where, $\varphi(t, t_0)$ is the creep coefficient and $E_{ci}(t_0)$ is the Young's modulus at a concrete age of 28 days. $\varphi_{bc}(t, t_0)$ is the basic creep and $\varphi_{dc}(t, t_0)$ is the drying creep.

3.2. FEA Modelling

According to the modeling code method previously described, the purpose of this project is to model the T-shaped pre-cast beam using three different levels of accuracy which allow having a better understanding of the Long-Term performance of the beam in a period of 5 years' time and comparing the numerous results, namely the deflection at mid-span, stress distribution, the contribution of creep and shrinkage and the losses of the prestress force in the tendons wires. AN interesting and possible future step is to compare also with real measurements data provided by the University of Natural Researches and Life Sciences of Vienna. Hence, the element has been modeled by using a 3D solid mesh, 2D plane stress and 1D beam element. 1D beam element is the simpler model in terms of accuracy and time consuming which has three degree of freedom allowing investigating mainly the global deflection of the element, the stress without differentiating between the generated stress on the top and bottom layer of the beam and the prestress losses. 2D plane stress and 3D brick element (5-6 DOF) are more accurate systems providing better results but on the other hand, the invested time on modeling is much greater than the 1D model.

As aforementioned, the chosen software for this analysis is DIANA. The software is provided with powerful software which allow the analysis of a large number of structure especially concrete one. Its simplicity and intuitive tools permitted to build up a quite complicated 3D model mainly due to the numerous reinforcement bars embedded in the cast, simplify the work for each model and modify, when necessary, the material or geometrical parameters through error-messages and brief description of the mistakes.

3.3. Loading modelling

Regarding the deflection due to the self-weight and pre-tensioning forces, a linear analysis has been also conducted with the aim of both evaluating the goodness of the models in terms of stiffness and chosen mass and checking comparing the obtained values to the Euler-Bernoulli theory for a simply supported beam.

Successively, once the linear results are assessed, the attention is brought to a non-linear phased analysis, where in addition to the dead and pre-tensioning load an external point load with a magnitude of 65 kN, applied to the steel load plate, is considered with the aim of investigating the long-term performance of the beam up to 5 years.

3.4. Time investment in the FE modelling

Another and important aspect which is unfortunately not commonly faced in the modern literature is assessed, namely the time invested modeling using finite element software. Therefore, it is evaluated the usefulness of each model in terms of time and the quality of the accuracy of the given results.

Therefore, in the application chapter the analysis will be structured as following:

- Description of the structure geometry
- Material and parameters used in the codes
- Structural modeling
- Loading
- Linear analysis method related to the Bernoulli-theory
- Phase nonlinear analysis and different load pattern for each phase

While, in the remaining chapters the linear and non-linear analysis results will be showed, plotted in tables and time-scaled graphs and successively discussed in terms of structural response and time invested.

4. Application

As aforementioned, the application used for this project is a prestressed reinforced concrete beam (Fig 19) that is available in the literature from a set of tests performed by the University of Natural Resources and Life Sciences, Austria (Alfred et al., 2017). More precisely, a set of beams have been testes and monitored with other proposes rather than understanding the long-term performance. Nevertheless, and taking into account the availability, also, of some long-term monitoring on these tests, this case has been selected by taking benefit of this. In other words, the availability of monitoring data on these beams is a significant input in the discussion of the FE model selection focussing on the long-term performance.

Figure 19 – T-shaped beam layout

4.1. Description of the structure

The structure under analysis is a simply supported reinforced concrete beam, with a total length of 5.00 m, composed by a T-shaped cross section, with a total height of 0.30 m and width of 0.14 m is slightly conical. The flange of the T-beam has a dimension of 1.50 m by 0.07 m thickness (Fig 20).

This beam is prestressed with a pressure of 898 MPa by means of 8 × 7 - wire 0.5 inch strands (ST 1570/1770). It is worthwhile to note that the eight wires at position (S2) along a length of 2.00 m, i.e. there is no bound between the concrete and the tendon.

Additionally, two reinforcement bars, with 20 mm in diameter, are located in the lower reinforcement layer and another six reinforcement bars, with 14 mm in diameter, are located in the upper reinforcement layer. The lower reinforcement layer was anchored using four horizontal rebars in U-bolt shape with a diameter of 12 mm per side. Additionally, 10 rebars in U-bolt shape, with 6 mm of diameter, were also mounted at a distance of 0.50 m to each other along the bar. The T-beam flange has orthogonal reinforcement of 8 mm in diameter at a distance to each other of 0.20 m in the longitudinal and transverse directions.

The supports are materialized by means of two cylinders, which are made of steel, and positioned at 12.5 cm from both end-sides of the element.

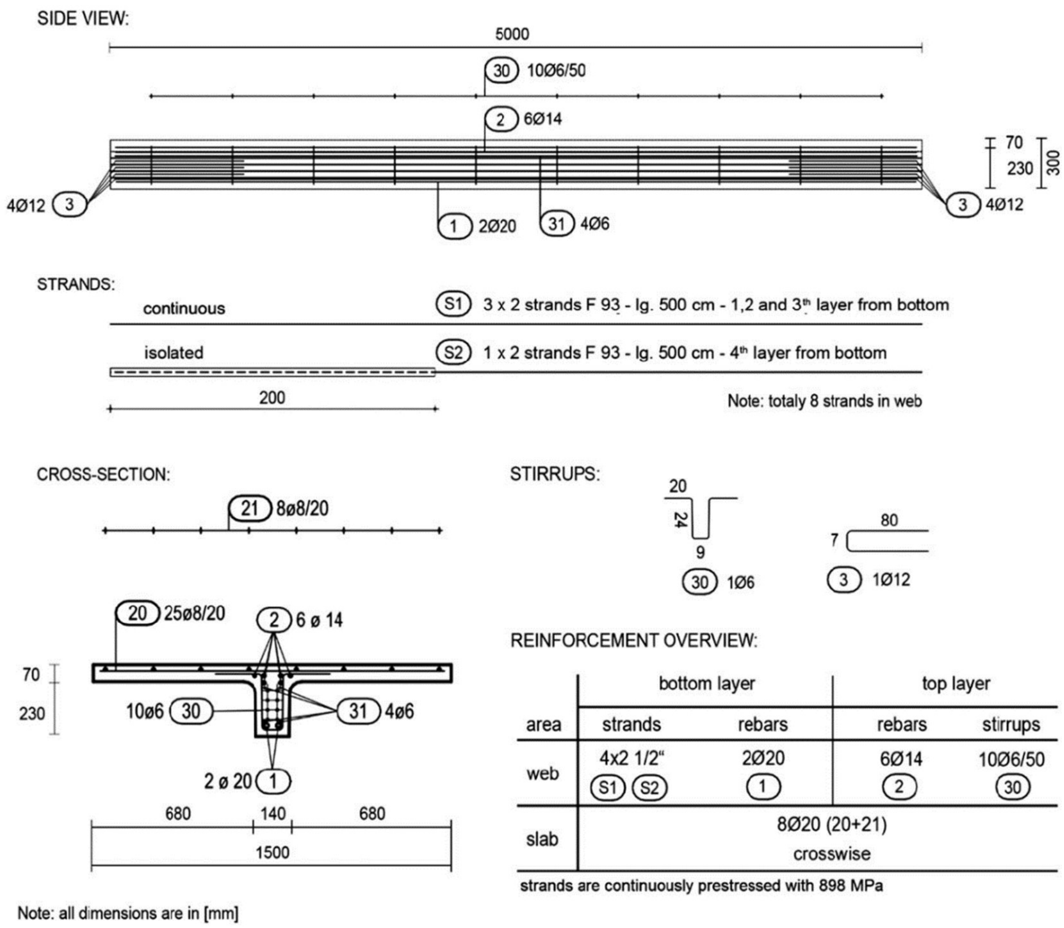


Figure 20 – Geometry and reinforcement of beam T30/150 V2 (Alfred et al., 2017)

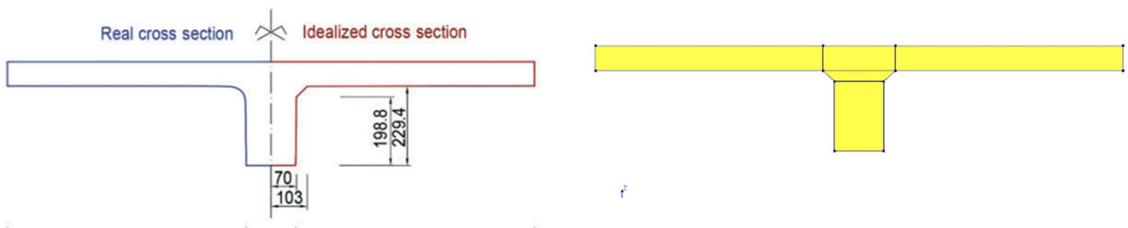
4.2. FE analysis

A nonlinear FE analysis was performed by using the general-purpose FE code DIANA (Manie 2008). Structural discretization was carried out using beam elements, plate elements and brick elements, depending on model approach. The ordinary and prestressed reinforcements were modelled using embedded reinforcing elements, whose deformation was calculated from the displacement field of the concrete FE in which they were embedded. Both instantaneous and time-dependent prestress losses were automatically computed.

The analysis considered two steps: (i) a linear static analysis to check the model stiffness and, by comparing the different model approaches, that the results are consistent and then (ii) a time-phased analysis was conducted to analyse the long-term performance in terms of displacements, strains and stresses.

4.2.1. 3D FE model

The 3D model (Fig 22) tries to simulate as much as possible the real shape of the element. The main difference is in the connection between the web and the flange where the section was idealized using a polygonal shape (Fig 21.a). The beam is composed by 5 main concrete elements which are a portion of the web, a trapezoid that portrays the connection of the web-flange, a central rectangular above the trapezoid as a useful guideline for the load plate and the two remaining portions of the flange (Fig 21.b).



a) section idealization **b) geometric modelling**

Figure 21 – Modelling approach for the T-beam cross-section

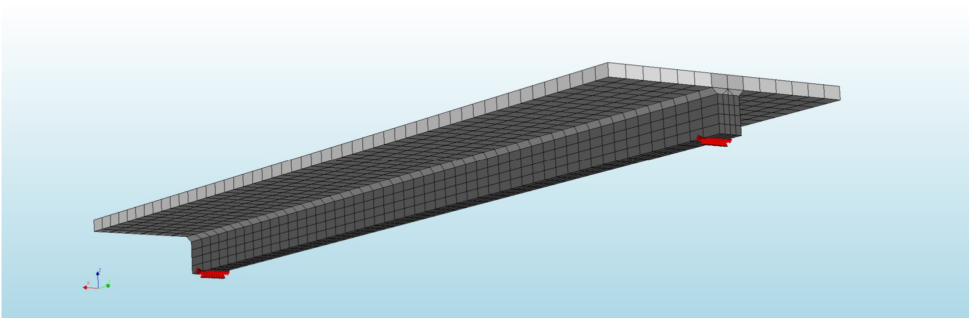


Figure 22 – 3D FE model of the T-beam (DIANA output)

As far as the reinforcement's layout is concerned, globally, the numerical model has 24 horizontal stirrups element, 49 vertical stirrups elements, 182 transversal reinforcement elements and 24 reinforcement bars element. Fig. 23 shows the layout obtained directly from DIANA software.

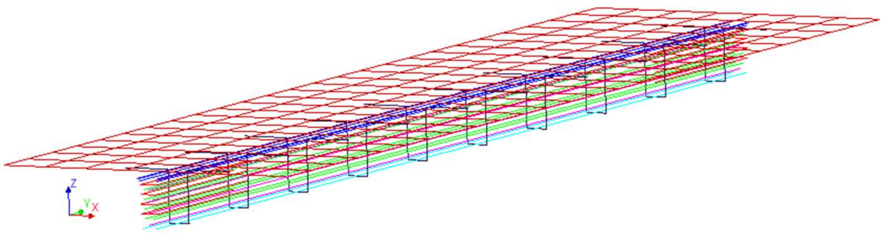


Figure 23 – 3D view of the reinforcement modelling (DIANA output)

4.2.2. 2D FE model

The 2D plane stress model is a simpler version of the previous 3D model. It does not consider the web-flange connection as a polygonal shape but as a perfect bond between the two rectangles. Therefore, the beam is composed by two rectangular concrete parts whit a thickness equal to their real width, namely 14 cm and 150 cm. Fig 24 shows the mesh of the 2D model elaborated under the scope of this work.

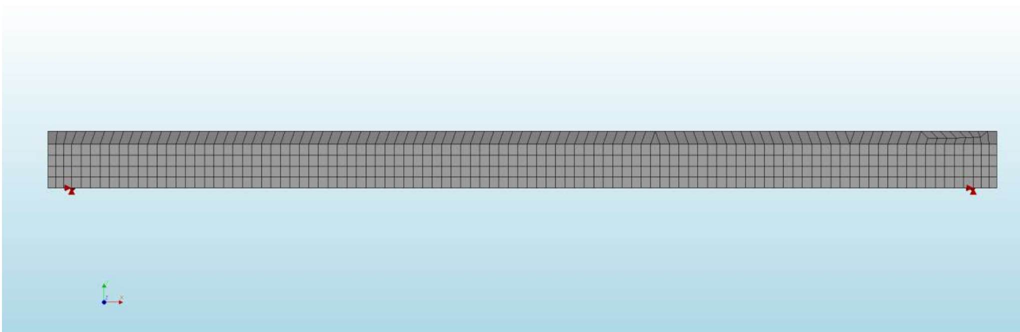


Figure 24 – 2D FE model of the T-beam (DIANA output)

Regarding the reinforcement bars, it was opted to maintain the analogue scheme, although with some slight simplifications in terms of layout. Consequently, it was given a different geometry to each bar comparing to the previous model due to the fact that in this case the modelling space is 2D. Hence, in order to calculate the area of the cross section it was taken into account the number of bars at the same height (i.e. 8 bars in the flange, the area is equal to the area of a single bar times 8 and so on). Fig 25 shows the layout obtained directly from DIANA software. Hence, it possible to see that this model has 8 horizontal bar elements, 8 horizontal stirrups per each end-side, 10 vertical stirrups elements and four tendons.

Figure 25 – 2D view of the reinforcement modelling (DIANA output)

4.2.3. 1D FE model

Finally, the third model implemented for this work is a 1D type (Fig 26). It is the simplest model that can be implemented, which only used linear elements. The reinforcement layout is the same layout as the one adopted for the 2D model. It is worth to say that the position of the beam element matches the barycentre of the cross section of the T-beam (Fig 21), i.e. considering the geometry of both bars and the T-shaped. For the cross-section definition, DIANA interface was used that easily allows to set up and edit the dimensional measures of a T-shape cross-section type.

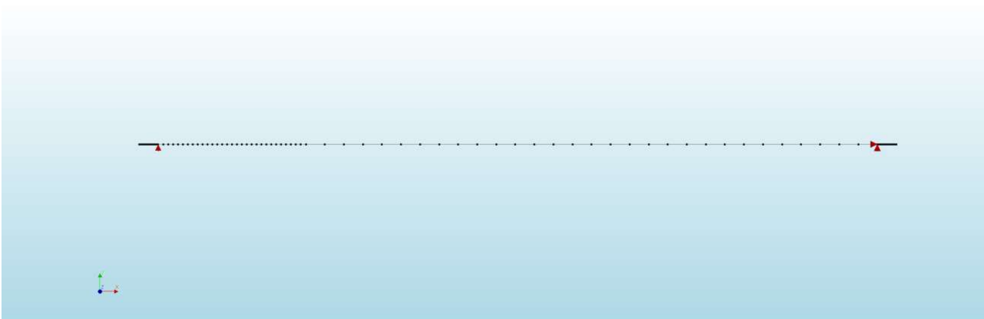


Figure 26 – 1D FE model of the T-beam (DIANA output)

4.3. Materials modelling

In concrete structures, to have a correct simulation and prediction of the structural behaviour a careful modelling of the material properties and parameters is necessary and required. The time-variation of the concrete is described by fib Model Code of structures 2010. The concrete class is C45 with a young's modulus of 37485.5 MPa and a density of 2.45e-06 Kg/mm^{2,} while the thermal expansion is 1e-05. The reinforcement bars are made of steel with a young's modulus of 200000 MPa and it was considered a bi-linear stress-strain graph with a yield stress equal to 450 MPa. Regarding the steel tendons wires, it was used two different material set up, in order to take into account, the prestress losses during the phase analysis, namely one is bonded to the mother element while the second one is not. Their young's modulus is 198000 MPa and the yield stress is 1875 MPa. The steel load plate has an elastic modulus of 210000 MPa and a Poisson ratio of 0.3. The following table summarizes all material properties.

Table 2 – Main mechanical properties of the materials

	Concrete	Steel bars	Steel tendons	Steel load plate
Young's Modulus E [GPa]	37.49	200	198	210
Compressive strength [MPa]	45	-	-	-
Tensile strength f_{ct} [MPa]	2.7	-	-	-
Medium Tensile strength [MPa]	3.8	-	-	-
Cement hardening (32.5 N)	Normal	-	-	-
Density [Kg/m^3]	2450	8050	8050	8050
Poisson ratio	0.2	0.3	0.3	0.3
Yield stress [MPa]	-	450	1875	-

4.4. Loading modelling

In this analysis many different loads have been taken into account, namely the self-weight of the beam, the prestress force in the tendons wire and a point load acting on the steel load plate. The self-weight of the reinforced concrete is 2500 Kg/m^3, the pre-tensioned force, acting in the four tendons, is equal to 898 MPa and the point load has a magnitude of 65 kN. As it will be discussed in the following sub-chapters, in the linear analysis only the prestress force and the self-weight will be considered. On the other hand, regarding the non-linear analysis, both dead loads, prestress and point load will be analysed and taken into account. Table 3 shows the loads type and the related magnitude.

Table 3 – Loading conditions

Load type	Magnitude	Analysis
Self-weight	3.36 kN/m	Linear/Phased
Steel bars	898 MPa	Linear/Phased
Point Load	65 kN	Phased

4.5. Boundary modelling

One of the key issues in the FE modelling was how to model the support conditions. In the laboratory test, steel cylinders were used as roller and pinned connections. Therefore, the boundary conditions can be assumed to lead to a simply supported beam. Hence, and independently of the model type (i.e. 1D, 2D or 3D). The support plate was reduced to a single point.

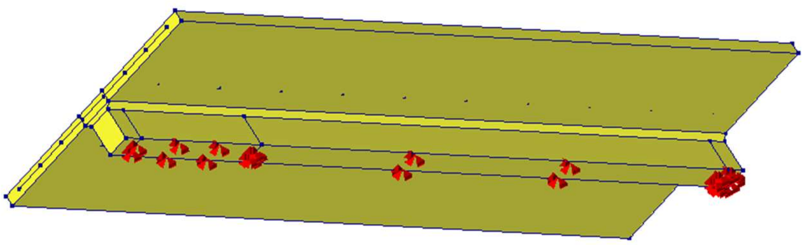


Figure 27 – Detail on the boundary conditions of the prestressed concrete beam (DIANA output)

4.6. Type of analysis

4.6.1. Linear analysis

A linear analysis has been conducted with the aim of evaluating the primary behaviour of the three models in terms of deflection due to the dead load and the prestressing force analysing and checking if the stiffness of the 1D, Plane stress and 3D model matched. This analysis is not a long-term investigation but a punctual one (t=0). Hence, two load combinations were used, considering separately the two load cases and the results, as we will see in the next chapter, were compared to the Bernoulli-linear elastic theory in order to be sure to have refined values.

4.6.2. Phased analysis

The nonlinear analysis is the main scope of this dissertation. The finite element software DIANA ran this analysis with the aim of investigating the behaviour of the beam in a temporal interval of 5 years (1825 days). Therefore, to achieve that the analysis has been divided into three main phases: i) Pouring period, ii) Pre-tensioning of the cables and iii) Application of the external load. The first phase has duration of 3 days where, ideologically, the beam is on the ground with no loads acting on it but the action of the creep and shrinkage start taking place. Therefore, another set of boundary conditions has been placed on the bottom face/layer of the web to simulate this aspect (Fig.27) and both the load plate and the supports are not installed on the element. The second phase is acting at the third day time where, the beam is positioned on the supports, previously described, and pre-tensioned with a force of 898 MPa and the self-weight start acting on it. At this stage the load plate is still not installed and the tendons are not bonded to the mother element. Successively, the last and third phase is the final stage when the element starts working and the period takes from 3 days until the end of the analysis, namely 5 years and the external point load is applied on the beam. Consequently, now the load plate is installed and loaded, the prestressed bars are now bonded to the mother element in order to take into account the losses of the pre-tensioning force due to creep and shrinkage and steel relaxation. The following table summarizes each phase components in terms of elements, loads and time.

Table 4 – Time history

	Phase 1 t∈[0, 3] days	Phase 2 t = 3 days	Phase 3 t∈[3, 1825] days
Self-weight	✓	✓	✓
Prestressing	✓	✓	✓
Point Load	-	-	✓
Supporting conditions	Fully supported on the ground	Simply supported	Simply supported

5. Results and discussion

As mentioned previously, any prediction obtained from FE models present differences to the real behaviour of the structure that is being modelled. Several reasons have been outlined for this fact, where one of them is related to the FE modelling approach for simulating the geometry and material properties. Therefore, in this section is summarized the main results obtained from the three FE models developed for the prestressed concrete beam. Firstly, an assessment of the models is done based on a static linear analysis and then, a set of comprehensive results are presented further in the time-phased analysis.

5.1. Linear elastic analysis

When a problem holds some complexity, as it is the case herein analysed, firstly, an assessment of the three models is promoted with the aim to assess/check that all inputs in the model is accurate and are coherent. For this, the structural response of the beam is evaluated for the three loads under analysis mainly: (i) self-weight, (ii) prestressing and (iii) punctual load. In order to support this, the analytical solution from the theory of Euler-Bernoulli is also calculated, mainly for the vertical displacement at the mid span section of a simply supported beam due to the self-weight (Eq. (10)).

$$\varphi = \frac{5 \cdot ql^4}{384 \cdot EI} \tag{10}$$

Where, q is a uniformly distributed load (in this case the beam self-weight), l is the beam length, E is the Young's modulus of elasticity and I is the inertia of the cross-section of the beam. Take into account that this calculation is mainly for assessing that the quality of the FE modelling approach, and for sake of simplicity, the self-weight has been quantified without considering the weight of the reinforcement bars. Hence, and according to the information aforementioned (Chapter 4), the following Table 5 summarizes the obtained results, from the FE modelling and the analytical solution, regarding the vertical displacement at the mid-span section of the beam.

In addition, Figures 28-30 shows the deflection shape obtained for the different loads considered.

Table 5 – Vertical displacements at the mid-span section (mm)

FE model type	Self-weight	Prestressing	Punctual Load	Analytical solution (for a UDL).
1D: Beam elements	-1.6	7.2	-7.8	-1.3
2D: Plane stress elements	-1.3	7.0	-6.1	
3D: Brick elements	-1.5	6.8	-6.3	

a) 1D model

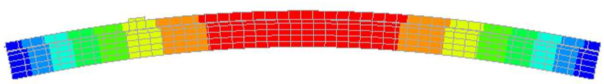
b) 2D model

c) 3D model

Figure 28 – Deflection of the beam due to the self-weight (DIANA output)

a) 1D model

b) 2D model

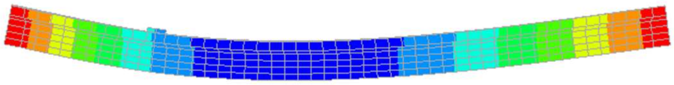


c) 3D model

Figure 29 Deflection of the beam due to the prestressing (DIANA output)

a) 1D model

b) 2D model



c) 3D model

Figure 30 – Deflection of the beam due to the punctual load (DIANA output)

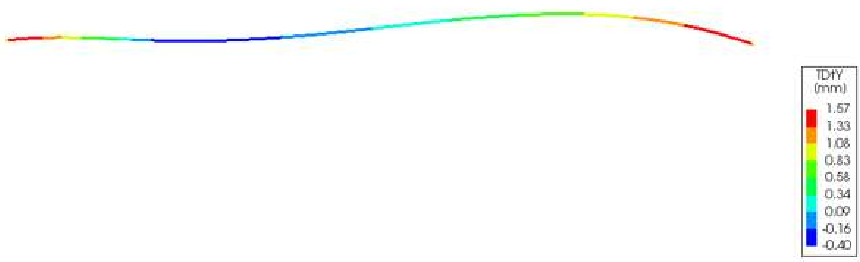
5.2. Time-phased analysis

Further to the linear static analysis previously performed, now the results obtained for the time-phased analysis is presented. More precisely, and according to what has been outlined in the previous section, the time-phased analysis consider three main phases, mainly: (i) Phase 0 (from t = 0 days to t = 3 days) corresponding to the period where the beam is poured and laying down on the floor until it reaches the required maturity in terms of strength; (ii) Phase 1 (t = 3 days) corresponding to the point in time where the beam is placed on the testing position (i.e. simply supported) and the prestressing is applied to both tendons and the self-weight is mobilized as well, and the external punctual load is applied on the steel load plate; (iii) Phase 2 (from t = 3 days to t = 1825 days) corresponding to the period where the time-dependent properties of concrete and prestress steel manifest their influence in the evolution of displacements and stresses.

Taking into account the focus of this work, and also the static linear analysis results presented in the previous section, only the results related to Phase 2 are presented. A comprehensive list of results can be found in the Appendix.

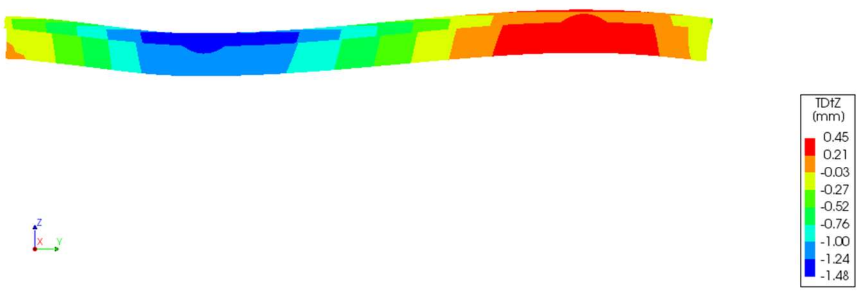
5.2.1. Beam deflection and stress diagrams for t = 1825 days

Below, results for the beam deflection (i.e. its shape) and the stress diagrams, along all the beam length, are presented for all FE models developed and presented in the previous chapter. It is worth to highlight that they are presented by groups of three in order to improve the results comparison between models.



a) 1D: Beam elements

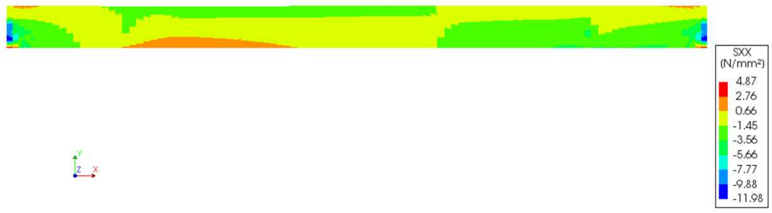
b) 2D: Plane stress elements



c) 3D: Brick elements

Figure 31 – Beam deflection at the end of 5 years (DIANA Output)

a) 1D: Beam elements



b) 2D: Plane stress elements

c) 3D: Brick elements

Figure 32 – Stresses on concrete at the end of 5 years (DIANA Output)

5.2.2. Time -series for displacements, stresses and forces from t = 3 days to t = 1825 days

The following time-scaled graphs will give a more detailed view about how the beam response changes over the years immediately after the tensioning phase. As previously mentioned, the 2D and 3D model show the differences between the top and bottom of the beam. DIANA calculates the stress and the strain of the 1D model at the barycentre of the cross section. Therefore, following the theory of De Saint Venant, the stress was reduced by 7% and plotted in the graph which shows the top layer response. While, regarding the bottom layer, due to simplicity reasons, it was decided to provide the 1D stress value in the following Table 8. The results are calculated at the mid span.

UNIVERSITY OF SURREY

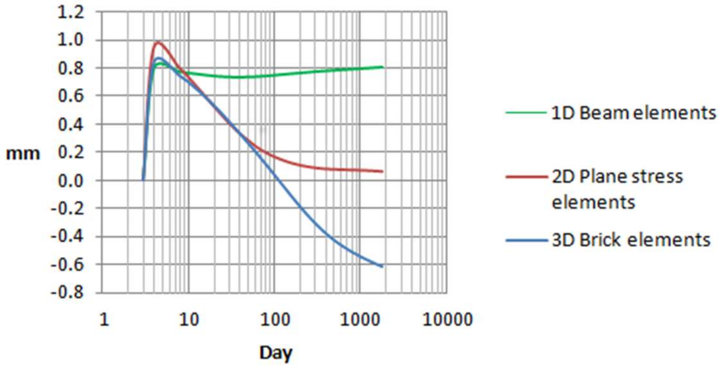


Figure 33 – Vertical displacement at the mid span section

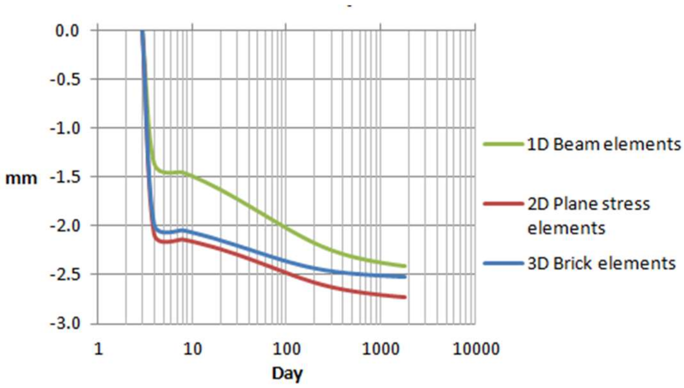


Figure 34 – Horizontal Displacement at the simply supported section

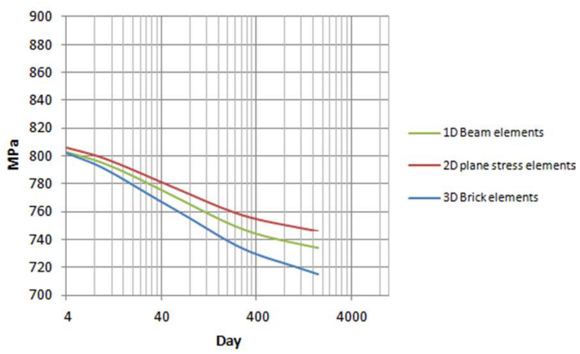


Figure 35 – Prestress Losses (DIANA Output)

Further to these graphical results, Table 6 to Table 9 aim to summarise the main indicators, accordingly, and by identifying two important points in time, i.e. t = 3 days, corresponding to the beginning of the long-term effects and at t = 1825 days (5 years), corresponding to the time-window considered for this analysis.

Table 6 – Vertical displacements at the mid-span section (mm)

FE model type	t = 3 days	t = 1825 days
1D: Beam elements	0.9	0.7
2D: Plate elements	1	0.1
3D: Brick elements	0.7	-0.6

Table 7 – Horizontal displacements at the simply supported section (mm)

FE model type	t = 3 days	t = 1825 days
1D: Beam elements	-1.4	-2.4
2D: Plate elements	-2.3	-2.7
3D: Brick elements	-2.1	-2.5

Table 8 – Stresses in the prestress tendons (MPa)

FE model type	t = 3 days		t = 1825 days	
	Tendon 1	Tendon 2	Tendon 1	Tendon 2
1D: Beam elements	811	811	737	732
2D: Plate elements	813	813	750	748
3D: Brick elements	813	813	720	718

5.3. Time invested on the FE analysis

The time invested behind modelling with finite element software is still an unknown argument of interest. As we have seen the analysis has been split into three principle levels of accuracy; 1D, 2D and 3D. Each model has many differences in terms parameters, geometry, difficulties in modelling and final numerical results. Therefore, the following analysis highlights the time expenditure of each model compared to the accuracy and the related displacement numerical values previously showed.

Table 9 – Numerical vertical displacement and invested time

FE model type	t = 3 days	t = 1825 days	Time invested	Run time*
1D: Beam elements	0.9	0.8	4 h	1 min
2D: Plate elements	1	0.1	1 Day	2 min
3D: Brick elements	0.7	-0.6	4 Days	7 min

* for 1 single analysis

5.4. Discussion on the results

5.4.1. Linear static analysis

Overall, it can be argued that there is a slightly difference between the three models and the analytical solution. Namely, the analytical analysis is based on the Euler-Bernoulli theory and the related formula (10) for a simply supported beam. For simplicity it hasn't taken into account the self-weight of the bars. Consequently, this is the reason why the analytical value is slightly lower than the 1D result due to the self weight. On the other hand, 2D and 3D model show closer results regarding the action of punctual load compared the 1D one which seems to be more flexible. This is mainly due to the distribution of the stress inside the cross section where both in the plane stress model and especially in the solid brick model there is a higher dissipation of the tension due to local deformations. However, the linear analysis provided evidence of the goodness of the results which are close and comparable allowing a good assessment of the stiffness and the mass and providing a useful tool for preparing and checking the three models for the non linear analysis.

5.4.2. Time-phased analysis

Clearly, the figures 34-38 provide information about the deflection and the stress distribution shape after 5 years time. It is clear how the pre-tensioning force influences the behaviour of the structure forming a sort of S-shape due to the presence of the point load acting on the steel plate in the shear area of the element.

It is interesting to note how the longitudinal stress is behaving like a rigid displacement between the top and the bottom layer; this is mainly due to the position of the mid-span compared to the flexural point in the beam. Notably, the 2D and 3D results, according to Figure 37, completely matched just after 3 days while at the end of the process they show a difference of longitudinal stress around 30 MPa. This behaviour can be noted for both top and bottom layer. This is may be mainly due to the approximation of the connection between the web and the top flange, where in the 2D there is a perfectly bond between two rectangles while in the 3D model the curvature is approximated with a polygonal shape.

In terms of prestress losses (Fig.36) it can be quantified an overall loss of 50-70 MPa , namely for the 1D model after five years the losses are about 70 MPa, while in the plane stress model shows a lower value up to 60 MPa. Notably, in the 3D brick elements model the pre-tensioning losses are up to 90 MPa, much greater than the other two measurements. Therefore, the 2D and beam model tend to underestimate the losses compared to the solid analysis with a difference approximately of 20-30 MPa.

Probably, the displacement is the most interesting and important graph because shows how the non linearity can change dramatically the behaviour of the element highlighting big differences between the three models. The 3D model, similarly to the linear results has immediately after three days the lower deflection at the mid-span (0.7 mm), while both the 2D model and 3D practically matched with a value, respectively of 1 mm and 0.9 mm. After 5 years, it is the 3D which shows the greater deflection up to -0.6 mm even if it had the lower hogging value (at 3 days) due to the tendons' pre-tensioning while the 2D practically does not deflect (0.1 mm) and 1D still shows a hogging curvature with a displacement equal to +0.8 mm. This is certainly due to how the stress is spread along the beam, the connection between the flange and the web, the complex relationship between the creep and shrinkage, the steel relaxation and the position of the embedded bars which in the one dimensional model and 2D are approximated by the equivalent area. Consequently, the real position in the space, for these two models, is neglected.

The strain behaviour, like the stress, has been divided into Top and bottom layer at mid-span section. Regarding, the top layer the 3 D model shows a greater strain after 3 day up to 250 micro strains, while after 5 years the difference is closer with a difference of 30 micro-strains from the 2D model but it matches with the beam element one. The 2D and 1 D practically match for the all process showing a slightly difference at the end of the 5 years. The bottom layer strain highlights the fact that the plane stress model and 3D one almost match showing a difference up to 30 micro strains at 1825 days.

Consequently, it can be argued that, meanwhile the liner analysis provide evidence that the three models are close in terms of deflection, the non-linear analysis shows big differences in terms of deflection and longitudinal stress. The differences between the 1D and 2D are closer for the vertical displacement case, while they notably differ from the 3D model, +0.8 mm and 0.1 mm compared to -0.6 mm respectively.

5.4.3. FE model accuracy vs. time investment

According to the collected results, it is clear how a different method of modelling can vary providing a wide spectrum of results that sometimes can match or at least be close with a good approximation or show notable differences. Hence, it is a duty of the engineer to choose the right method of simulation. Namely, regarding the aforementioned analysis, we have seen how the type of analysis influences the final results (i.e. Linear and no-Linear). Clearly, the 3D solid model is for sure the most refined method that manages to better simulate the real behaviour of the T-shaped beam but on the other hand it took full four days to be modelled, according to Table 10. While, the 1D beam elements accuracy and the planes stress allow obtaining the final results in less than half time that was taken to complete the full solid model. Consequently, if we imagine expanding this accuracy concept to a full-scale structure the time needed to complete a full and refined solid model would increase dramatically and nowadays time has an impact on the costs. As a consequence, firstly, the choice of the proper accuracy methodology depends on the type of the structure (Bridge, slabs, pipes etc), the required performance and economic regulations in terms of completion of the works, payments of professional and expert figures who are able to use, calibrate and assess the finite element software results with real measurements taken on the field etc.

Hence, the assessment of the Long-term performance is a very important field nowadays and the right choice of the most suitable accuracy FE method has to be taken into account wisely.

6. Conclusions and further steps

The presented work focussed on the assessment of the long-term performance of a prestressed concrete T-shaped beam. The FEA software package DIANA has been used, which is one of the most well-recognized software packages to perform this type of analysis. Three types of FEA models have been considered, mainly: (i) 1D model with Timoshenko beam elements, (ii) 2D model with plane stress elements and (iii) 3D model with brick elements. Also, the time invested in developing an accurate FEA model was assessed in order to better decide which approach is more suitable in a cost-benefit basis. Based on the results obtained, the author consider the following conclusions of high relevance, mainly:

1. The analysis performed both a linear-elastic and non-linear analysis with the aim of highlighting the main difference in terms of the response of the three models. The linear analysis results showed the goodness of the chosen parameters and that the models stiffness was in agreement among the three FE models developed. Moreover, the outputs show how the vertical displacements of the beam in all models are close with differences of 6% due to the self weight and prestressed force and 20% between the two and 3D model compared to 1D model.
2. On the other hand, the long-term performance results show very interesting results. The behaviour of the models tend to differ notably regarding the vertical displacement, namely the 3D, which is the only FE model in which at mid-span the cross section deflects negatively, while in the other two models slightly deflects upwards. It seems that reducing the accuracy of the model tends to amplify this behaviour. This could be mainly explained due to the different notional sizes (i.e. the ratio of the volume by the exposed perimeter) used for the three FE models. Indeed, this is the only parameter that is changed between the three FE models. Moreover, the results from the linear static analysis also indicates this, since that for this type of analysis the notional size parameter is not relevant, but it is determinant for the long term performance.
3. The strain response at the mid-span bottom layer offers a good simulation with differences of 4% while at the top layer the variation increases up to 11%. The higher difference in the top can be explained by the higher level of

the concrete stress, which allied to the notional size effect might potentiate the differences.

4. The stress behaviour at mid-span has the characteristic to be similar with the sign compared to the bottom layers. This is mainly due to the coupled-action of both pre-tensioning forces and the magnitude of the point load which induce on the beam a flexural point located at 2.6 m from the load plate. Therefore, at mid-span there is a zone which is almost flat; hence this is why the distribution of the stress presents small compressive values on both layers.
5. The prestress losses do not show significant differences between different FE approaches, with differences going up to 4 %.
6. The time invested in the FE modelling revealed to be hugely different among different approaches. Especially, the 3D FE model requires almost 50% more additional work than the respective 2D FE version and and 80% more than the 1D FE version based on beam elements. Moreover, the time calculation is also completely different when extracting results with the 3D FE model taking ×7 time more time than the respective 1D FE version. In this context, a full 3D model for a full-scale bridge (and small-scale model as herein analysed that already takes 7 min to run a single time-phased analysis) is only viable if the criticality and the socio-economic impact of a potential malfunctioning of the bridge might involve litigation issues.
7. Further comparison of these results to the available monitoring data is planned already. Indeed, the author is collaborating on this envisaging a joint-publication with his supervisor and the colleagues from the University of Natural Resources and Life Sciences, Austria, towards the next 12th international DIANA users meeting to be held in Porto, Portugal next October 2018.

References

[1] B. Torres, I. Payá-Zaforteza, P. A. Calderón, and J. M. Adam, "Analysis of the strain transfer in a new FBG sensor for Structural Health Monitoring," *Eng. Struct.*, vol. 33, no. 2, pp. 539–548, 2011.

[2] C. Rodrigues, C. Félix, A. Lage, and J. Figueiras, "Development of a long-term monitoring system based on FBG sensors applied to concrete bridges," *Eng. Struct.*, vol. 32, no. 8, pp. 1993–2002, Aug. 2010.

[3] Q.-J. Wen, "Long-term effect analysis of prestressed concrete box-girder bridge widening," *Constr. Build. Mater.*, vol. 25, no. 4, pp. 1580–1586, Apr. 2011.

[4] A. Strauss, B. Krug, O. Slowik, and D. Novak, "Combined shear and flexure performance of prestressing concrete T-shaped beams: Experiment and deterministic modeling," *Struct. Concr.*, vol. 19, no. 1, pp. 16–35, 2018.

[5] T. Tong, Z. Liu, J. Zhang, and Q. Yu, "Long-term performance of prestressed concrete bridges under the intertwined effects of concrete damage, static creep and traffic-induced cyclic creep," *Eng. Struct.*, vol. 127, pp. 510–524, Nov. 2016.

[6] H. Sousa, J. Bento, and J. Figueiras, "Construction assessment and long-term prediction of prestressed concrete bridges based on monitoring data," *Eng. Struct.*, 2013.

[7] T. Guo, Z. Chen, S. Lu, and R. Yao, "Monitoring and analysis of long-term prestress losses in post-tensioned concrete beams," *Measurement*, vol. 122, pp. 573–581, Jul. 2018.

[8] T. Sussman, "Solid and shell elements."

[9] CSI, "SAP2000. Analysis Reference Manual," *CSI: Berkeley (CA, USA): Computers and Structures INC*. 2016.

[10] "Analysis Manual."

[11] "DIANA Finite Element Analysis User's Manual Concrete and Masonry

Analysis Release 9.4." [Online]. Available: https://dianafea.com/manuals/d94/ConcMas/ConcMas.html. [Accessed: 04-Jul-2018].

[12] "EN 1992-1-1: Eurocode 2: Design of concrete structures - Part 1-1: General rules and rules for buildings."

[13] Laura, "Model Code 2010 Final draft," 2012.