

Politecnico di Torino

Master's Degree Course
In Building Engineering



**Politecnico
di Torino**

Master's Thesis

Three-Dimensional Nonlinear Finite Element Analysis of an Existing Reinforced Concrete Bridge Beam

Supervisor:
Prof. Diego Gino

Candidate:
Sepideh Yousefi

Co-supervisors:
Prof. Paolo Castaldo
Eng. Giovanni Francesco Bono
Prof. Elena Miceli

Academic Year 2025/2026

Abstract

A large part of the Italian road network relies on reinforced concrete bridges built in the decades after the Second World War. Many of these structures are now close to or beyond their intended service life, and several show clear signs of material degradation. Their condition has made structural assessment a priority, and the 2020 Italian Guidelines for existing bridges set out a multi-level procedure to carry it out. The most in-depth stage of this procedure is the Level 4 assessment, which relies on advanced analysis. Here nonlinear finite element analysis is the main tool, but its reliability rests on calibration against experimental data. Load tests taken to collapse on full-scale beam are rare, and they offer a valuable opportunity for such calibration.

This thesis builds and validates a three-dimensional nonlinear model of a 25.9 m reinforced concrete edge beam, removed from a decommissioned viaduct on the A6 motorway. The beam had been in service for about sixty years and showed clear degradation, including corrosion of the reinforcement and spalling of the cover. In July 2021, it was loaded to collapse at a dedicated test field. The load, made of cast iron ballast, was applied in successive loading and unloading cycles arranged in four phases, to capture the progressive loss of stiffness. Collapse was in the fourth phase, during placement of the third layer (SC), at a total load of 1260kN.

The model was built in ATENA Engineering, using the ATENA studio interface. Its constitutive models rely on the destructive tests carried out after the experiment, that is compressive strength of the concrete cores and the mechanical properties of the reinforcement samples. The remaining parameters of the concrete law were generated from these strengths through the *fib* Model Code 2010. Concrete was represented by a fracture-plastic law with rotating cracks. Four concrete zones were defined, and the lower strengths assigned to the collapse region were taken from statistical analysis of the concrete cores, which showed locally reduced values there. As a result, the model develops failure in the same region as the real beam. The reinforcement was modelled with discrete bars and a single cyclic steel law, and the load was applied in force control.

Because the cross-section is the asymmetric edge element of the original deck, the response of the supports has a strong effect on the measured behavior. For this reason, the bearing and the lateral restraint were not idealized, but studied through parametric analysis. Different material models, stiffness values and restraint conditions were compared, and the study led to two final models. In the first, the neoprene bearing is represented by a steel material with an equivalent

modulus of 5 MPa, without any added lateral restraint. In the second, the bearing is replaced by a Mohr-Coulomb interface with an equivalent modulus of 3.5 MPa, together with a transverse spring of 12 MPa that reproduces the lateral restraint.

Both final models closely follow the experimental response through the loading and unloading cycles, up to collapse, and they match the measured failure load. The work shows that a nonlinear model calibrated against the test, together with a realistic description of the supports, can reproduce the response of an existing RC beam after many years of working life exposed to the environment. The result supports the use of nonlinear finite element analysis in the Level 4 assessment of the existing reinforced concrete bridges.

Contents

Chapter 1.	Safety assessment of bridges.....	1
1.1	Introduction.....	1
1.2	Regulatory framework	3
1.2.1	Evolution of Italian legislation regarding bridge design.....	3
1.2.2	Guidelines for existing bridges	11
1.2.3	The European framework: Eurocodes and existing structures.....	14
1.2.4	The international framework: fib Model Code and JCSS	16
1.3	Safety formats for nonlinear finite element analysis.....	17
1.4	Sources of uncertainty in nonlinear finite element analysis.....	23
Chapter 2.	Load test on the reinforced concrete beam	25
2.1	Introduction.....	25
2.2	Test field	26
2.3	Geometry and reinforcement	27
2.3.1	Cross-section.....	27
2.3.2	Ordinary reinforcement.....	29
2.4	Materials and degradation.....	31
2.4.1	Design materials.....	31
2.4.2	Observed degradation	32
2.5	Load test execution	33
2.5.1	Test setup	33
2.5.2	Loading procedure	34
2.6	Monitoring systems.....	36
2.6.1	Topographic targets.....	37
2.6.2	LVDT extensometers	38
2.6.3	Strain gauges	39
2.7	Load test results	39
2.7.1	Collapse mode.....	40
2.7.2	Topographic measurements	41

2.7.3	LVDT measurements	45
2.7.4	Reinforcement strains	49
2.8	Laboratory tests.....	50
2.8.1	Statistical method.....	50
2.8.2	Concrete cores.....	51
2.8.3	Reinforcing steel	53
2.9	Summary	54
Chapter 3.	Nonlinear finite element model of the beam.....	55
3.1	Introduction to nonlinear analysis.....	55
3.2	Steps of the NLFEA.....	56
3.3	Solution strategy	57
3.4	The ATENA Engineering software package.....	58
3.5	Model geometry	61
3.5.1	Geometry of the beam.....	61
3.5.2	Crossbeams and slab	63
3.5.3	Supports and load distribution	64
3.5.4	Ordinary reinforcement.....	65
3.6	Constitutive models of materials.....	65
3.6.1	Concrete	66
3.6.2	Reinforcing steel	70
3.6.3	Elastic materials for plates and bearings.....	71
3.7	Creating and assigning structural elements.....	72
3.8	Creating and assigning ordinary reinforcement	73
3.9	Finite element mesh definition.....	74
3.10	Boundary conditions and modelling of the supports: the parametric study	75
3.10.1	Variants of bearing material.....	77
3.10.2	Interface element in place of the bearing	78
3.10.3	Steel plates modelled as elastic material with $E = 200 \text{ GPa}$	80
3.10.4	Lateral restraint from the third cycle: spring stiffness study.....	81

3.11	Load history	82
3.12	Monitoring points.....	85
3.13	Solution method and convergence criteria.....	87
Chapter 4.	Model variants and calibration: the parametric study	90
4.1	Introduction and approach	90
4.2	Stage 1: the bearing as a physical volume	90
4.3	Stage 2: the bearing replaced by an interface.....	91
4.4	Stage 3: the lateral restraint as a spring.....	92
4.5	The two final models.....	93
Chapter 5.	Comparison of the final models with the experiment, and conclusions.....	95
5.1	Deformed shape and failure mode	96
5.2	Displacement at the topographic targets	100
5.3	Rotation comparison	106
5.4	Reinforcement strain at the gauges	113
5.5	Conclusions.....	115
Bibliography		118

Figure

Figure 1.1. Traffic load schemes prescribed by Normale n. 8 of 15 September 1933 [7].	4
Figure 1.2. Traffic load schemes prescribed by Circular n. 6018 of 9 June 1945 [8].	5
Figure 1.3. Traffic load schemes prescribed by ANAS Circular n. 820 of 15 March 1952 [10].	6
Figure 1.4. Traffic load schemes prescribed by Ministerial Circular n. 384 of 14 February 1962 [11].	7
Figure 1.5. Traffic load schemes prescribed by the Ministerial Decree of 2 August 1980 [12].	8
Figure 1.6. Traffic load schemes prescribed by the Ministerial Decree of 4 May 1990 [13].	9
Figure 1.7. Traffic load schemes prescribed by the current NTC 2018 [15] aligned with EN 1991-2 [17].	10
Figure 1.8. Schematic representation of the multi-level approach of the 2020 Italian Guidelines for existing bridges [3].	13
Figure 1.9. Conceptual difference between the local approach (left) typical of linear-elastic analysis and the global approach (right) typical of NLFEA. Adapted from Allaix et al [19].	18
Figure 1.10. Relationship between the reliability index β and the probability of failure P_f , adapted from Sýkora et al. [23].	22
Figure 2.1. Layout of the test field at the former motorway service area.	27
Figure 2.2. Cross-section of the original deck with the tested edge beam highlighted.	28
Figure 2.3. Geometry of the 25.90 m beam.	29
Figure 2.4. Reinforcement layout of the beam and of the slab.	30
Figure 2.5. Main types of degradation observed on the beam.	33
Figure 2.6. Portion of the slab removed in the central region.	34
Figure 2.7. Ballast unit and arrangement of the loading layers.	36
Figure 2.8. Load history, as total load against a) loading step and b) against time.	36
Figure 2.9. Layout of the 24 topographic targets.	38
Figure 2.10. Layout of the LVDT extensometers in the monitored sections.	39
Figure 2.11. Position of the strain gauges SGSX and SGMZ on the bottom chord.	39
Figure 2.12. Reference system for displacements and rotations.	40
Figure 2.13. Photograph representation of the collapse by crushing of the slab concrete near the midspan.	41
Figure 2.14. Displacements of the target pairs against the loading step.	43
Figure 2.15. Flexural rotations in the five monitored sections for each 4 phases.	44
Figure 2.16. Torsional rotations in the five monitored sections for each 4 phases.	45
Figure 2.17. Longitudinal strain profiles at section $z=7.5\text{m}$, over the height of the monitored sections for all 4 phases.	46
Figure 2.18. Longitudinal strain profiles at section $z=12.2\text{m}$, over the height of the monitored sections for all 4 phases.	47

Figure 2.19. Longitudinal strain profiles at section $z=13.4\text{m}$, over the height of the monitored sections for all 4 phases.	48
Figure 2.20. Longitudinal strain profiles at section $z=16.8\text{m}$, over the height of the monitored sections for all 4 phases.	49
Figure 2.21. Strains measured by SGSX and SGMZ during the test.	50
Figure 2.22. Location of the 14 concrete cores.	52
Figure 2.23. Location of the sampling regions B1, B2 and B3.	53
Figure 3.1. Idealization of the structure into a mechanical model and discretization into a finite element model.	57
Figure 3.2. Graphical interface of ATENA Pre, adapted from the user's [44].	60
Figure 3.3. Graphical interface of ATENA Studio, adapted from the user's manual [45].	60
Figure 3.4. Geometry of the beam prepared in AutoCAD before the export to DXF.	62
Figure 3.5. Imported model of joints and lines into ATENA Pre, which representing the beam and the auxiliary elements of the test.	62
Figure 3.6. Imported lines into ATENA Pre, representing the ordinary reinforcement.	63
Figure 3.7. Model of the beam after the generation of the solid volumes in ATENA Pre.	63
Figure 3.8. Scheme of the support region as reproduced in the model.	64
Figure 3.9. Model of the beam reinforcement and stirrups in dedicated layers.	65
Figure 3.10. Material properties of the precast beam concrete.	67
Figure 3.11. Material Properties of the concretes in web assigned to the collapse region.	68
Figure 3.12. Material Properties of the concretes in top part of the collapse region.	69
Figure 3.13. Definition of the reinforcement material from the experimental stress-strain function. ...	71
Figure 3.14. Definition and assignment of the structural element types.	72
Figure 3.15. Reinforcement of the model after the assignment of the bar types.	73
Figure 3.16. Finite element mesh of the model, global size 0.20 m along the beam axis.	75
Figure 3.17. Load cases defined in the model.	76
Figure 3.18. Support boundary condition at the base of the plinths.	77
Figure 3.19. Definition of a) Steel and b) Elastic materials for bearing.	78
Figure 3.20. Failure criterion of the CCInterface element [44].	80
Figure 3.21. Lateral restraint of the test and its idealization in the model, a) fully restraint; b) additional transversal spring.	82
Figure 3.22. First interval of the load history.	85
Figure 3.23. Typical loading interval of the third and fourth phases.	85
Figure 3.24. Monitoring points defined in the model.	87
Figure 3.25. Scheme of the Newton-Raphson method [41].	88
Figure 5.1. deformed shape at maximum deflection. (a) real beam at collapse; (b) steel model; (c) interface model.	97

Figure 5.2.Deformed shape at maximum deflection, side view. (a) steel model; (b) interface model.	98
Figure 5.3.Deformed shape at maximum deflection, transverse view. (a) steel model; (b) interface model.....	98
Figure 5.4.crack pattern of a) the steel model and b) the interface model at the load step near collapse. The 45-degree shear cracks near the supports and the bending cracks at mid-span.	99
Figure 5.5.vertical displacement at P9 and P10, against the loading step and against the total load, experiment against a) Model with steel and b) Model with interface.	101
Figure 5.6.transverse displacement at P9 and P10, against the loading step and against the total load, experiment against a) Model with steel and b) Model with interface.	102
Figure 5.7. vertical, transverse and longitudinal displacement at targets P1, P2, P6, P7, P8, P9, P10 and P1', P2', P6', P7', P8', experiment against Model with steel.	104
Figure 5.8.vertical, transverse and longitudinal displacement at targets P1, P2, P6, P7, P8, P9, P10 and P1', P2', P6', P7', P8', experiment against Model with interface.	106
Figure 5.9. rotation about the transversal axis (flexural rotation) at five sections along the span, for the four loading phases, a) experiment against b) the FEM analysis of the model with steel as bearing material.....	108
Figure 5.10. rotation about the transversal axis (flexural rotation) at five sections along the span, for the four loading phases, a) experiment against b) the FEM analysis of the model with Interface.	109
Figure 5.11.rotation about the longitudinal axis (torsion) at five sections along the span, for the four loading phases, a) experiment against b) the model with steel as bearing material.	111
Figure 5.12.rotation about the longitudinal axis (torsion) at five sections along the span, for the four loading phases, a) experiment against b) the model with interface.....	112
Figure 5.13.reinforcement strain at SGSX (third point) and SGMZ (mid-span) against the loading step, experiment against a) Model with Steel material and b) Model using interface.	114

Tables

Table 1.1.Evolution of the Italian regulatory framework for the design of road bridges. The table includes the most relevant standards from 1933 to today.	11
Table 2.1 .Design properties of the concrete assumed from the original documentation, for the cast-in-place slab and crossbeams (class 250) and for the precast beam (class 350).	31
Table 2.2.Load history of the test summarized by loading cycle, with the ballast layers, the loading steps and the peak total load of each cycle.	35
Table 2.3. Statistical parameters of the in-situ cylindrical compressive strength of the concrete, with the elastic moduli measured on cores A and B.	52
Table 2.4.Statistical parameters of the ordinary reinforcing steel obtained from the tensile tests.	53
Table 3.1.Concrete materials of the model: mean compressive strength and elastic modulus.	66
Table 3.2.Mean properties of the reinforcing steel from the tensile tests, assigned to all bar groups. .	70
Table 3.3.Composition and shape quality of the finite element mesh (86,983 nodes).	75
Table 3.4.Load history of the model: 49 intervals and 106 steps reproducing the experimental loading program.	83
Table 3.5 .Convergence tolerances adopted in the analyses (program defaults).	89
Table 4.1.Configurations tested in the parametric study.	93
Table 4.2.Parameters of the two final models.	93
Table 5.1.The two final models carried forward from Chapter 4.	95

Chapter 1. Safety assessment of bridges

1.1 Introduction

In the last few years, the structural safety of existing bridges has become one of the most debated engineering topics in Italy. Several factors contribute to this growing attention. A large portion of the Italian infrastructure was built in the second half of the 20th century, so many bridges are now close to the end of their original service life, or have already passed it. At the same time, the weight and frequency of traffic loads have increased substantially compared to those considered in the original design, and several serious failures in recent years have brought the problem to the attention of both the public and the engineering community.

The collapse of the Polcevera Viaduct (also known as the Morandi Bridge) in Genoa, on 14 August 2018, caused 43 fatalities and marked a turning point in how bridge safety is perceived in Italy [1], [2]. Less than two years later, on 8 April 2020, the Albiano Magra bridge between Liguria and Tuscany collapsed without casualties, only because the traffic was very limited due to the pandemic lockdown. These two events pushed the Italian government to revise the national approach to the management of the existing bridge stock. The most important technical outcome of this revision is the publication, in 2020, of the *Linee Guida per la classificazione e gestione del rischio, la valutazione della sicurezza ed il monitoraggio dei ponti esistenti* (Guidelines for the classification and management of risk, safety assessment and monitoring of existing bridges), adopted with Ministerial Decree n. 578 of 17 December 2020 and updated in 2022 [3], [4].

Outside of Italy, the same trend can be observed at the European level. The current Eurocodes already include rules for existing structures, and the next revision of EN 1990 will extend its scope to the verification of existing works in a more explicit way. At the international level, the fib Model Code 2010 and the more recent fib Model Code 2020 both dedicate full chapters to the assessment of existing concrete structures and to the role of nonlinear finite element analysis (NLFEA) as the most accurate level of analysis available to the engineer.

The assessment of an existing bridge is a different problem from the design of a new one. Geometry and materials are not freely chosen by the engineer, but are fixed by what was built and by how the structure has aged. Mechanical properties and structural response change with time because of several phenomena that act together [5]. The main ones are:

- the time-dependent behavior of the materials, including creep, shrinkage of concrete and relaxation of prestressing steel where present;
- environmental degradation, mainly carbonation and chloride penetration of concrete, which in time produce corrosion of the reinforcement and spalling of the cover;
- the cumulative effect of cyclic traffic loads, which contributes to fatigue damage of concrete and reinforcement and to the progressive opening of existing cracks;
- the historical increase of the live loads carried by the structure, which in some cases are well above the values used at the design stage.

Another important point is the cost of any intervention on an existing bridge. Strengthening, retrofitting or replacing a bridge is expensive in economic, social and environmental terms. These works produce significant CO₂ emissions, require traffic interruptions and consume a large part of the public budget available for infrastructure maintenance. For this reason, it is essential to identify with good accuracy the bridges that really need an intervention, avoiding both the underestimation of risk, with the consequences on safety, and its overestimation, with the consequences on cost. NLFEA finds its main engineering value in this trade-off, because it takes into account the real nonlinear behavior of the materials and the redistribution of internal forces, NLFEA usually predicts a global resistance closer to the real one. The estimated capacity is often higher than the one obtained from a conventional linear-elastic analysis, and this allows interventions to be limited to the cases where they are really necessary.

Following this approach, this thesis develops a three-dimensional nonlinear finite element model of a reinforced concrete bridge beam taken from a decommissioned viaduct. The model is calibrated against the results of an experimental load-to-collapse test performed on the same beam [6]. The work is presented as a representative example of a Level 4 in-depth assessment according to the 2020 Italian Guidelines [3], and as a model that can be reused for similar reinforced concrete bridge girders built in the same period.

1.2 Regulatory framework

The Italian regulatory framework for bridges has been built up over almost a century, through several standards that reflect the engineering and social context of their time. Understanding this evolution is not only a historical exercise, it has a direct practical importance for the assessment of existing bridges, because the verification of an existing structure must start from the identification of the design loads and the safety criteria used when the bridge was built. Section 1.2.1 follows this evolution from the first national standard of 1933 up to the current NTC 2018 and the related 2019 Circular. Section 1.2.2 then introduces the 2020 Linee Guida, which add to the NTC a multi-level methodology developed specifically for existing bridges.

1.2.1 Evolution of Italian legislation regarding bridge design

Until the 1930s, the design of road bridges in Italy was not regulated by any uniform national standard. The first systematic technical document was published in 1933, the *Normale n. 8* of 15 September, issued by the Ministry of Public Works with the title *Carichi da considerarsi nel calcolo dei ponti per strade ordinarie* [7]. This standard divided the roads into three categories: major roads (state roads and motorways), medium-traffic roads (provincial roads), and low-traffic roads (municipal roads). For each category, a specific set of traffic loads was prescribed for design. For the first category, three alternative combinations had to be checked. The first one used two parallel indefinite columns of 12-tonne trucks together with a 400 kg/m² crowd load on the sidewalks. The second one used a single column of trucks with a different transverse position and the same crowd load. The third one used only the crowd load uniformly distributed over the deck. For the second category of roads, in addition to the first combination just described, two more configurations were prescribed. The first one used an indefinite column of 12-tonne trucks flanked by two 18-tonne steam rollers, with the crowd load applied on the remaining surface. The second one used two 18-tonne steam rollers placed side by side, again with the crowd load on the remaining area. For the third category, the combinations described above were revised to reduce the static load on the bridge. A separate 18-tonne steam roller scheme was used for the verification of the crossbeams and of the slab. The dynamic amplification of the loads was treated through a constant multiplier equal to 1.25, which applied to all three categories.

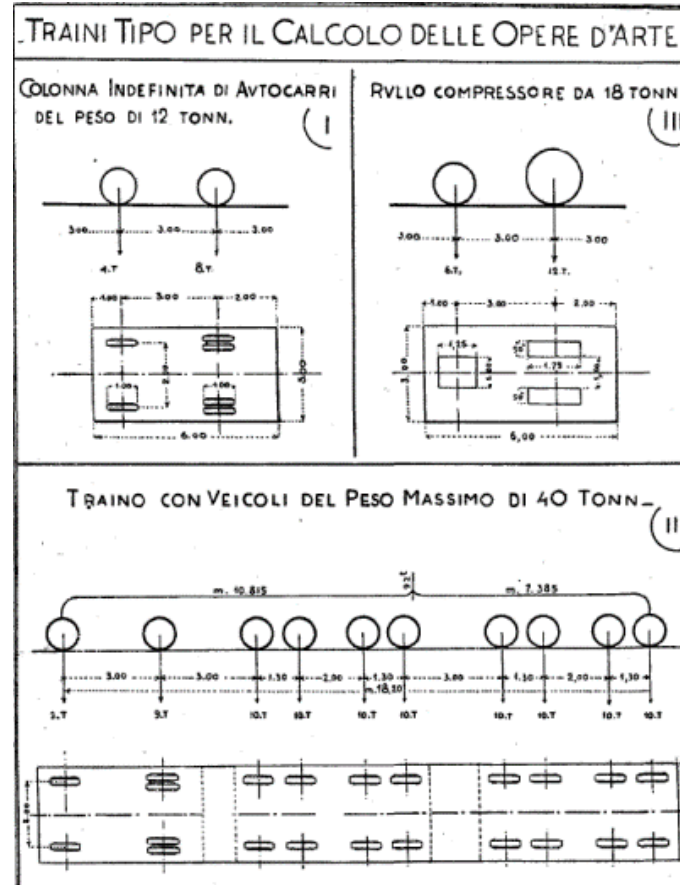


Figure 1.1. Traffic load schemes prescribed by Normale n. 8 of 15 September 1933 [7].

After the Second World War, the Ministry of Public Works updated the standard through Circular n. 6018 of 9 June 1945 [8]. The new document reduced the road classification to two categories, heavy-traffic and medium-to-light traffic roads, and removed the second of the load configurations described above. For heavy-traffic roads, two combinations remained. The first one used two or more parallel indefinite columns of 12-tonne trucks plus the 400 kg/m² crowd load on the sidewalks. The second one used two parallel 18-tonne steam rollers together with the same crowd load. The most important novelty of this circular was the introduction of a dynamic amplification coefficient that depends on the span of the bridge for medium-to-light traffic:

$$\varphi = 1 + 16 / (L + 40) \quad (1.1)$$

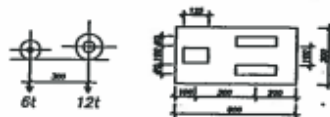
where L is the span in meters. This formula reflects the fact that shorter spans are more sensitive to dynamic effects than longer ones, which is a property that all later Italian and European codes have kept in different forms [9].

SCHEMI DI CARICO CIVILI

SCHEMA 1 - Colonna indefinita di autocarri da 12t



SCHEMA 2 - Rullo compressore isolato da 18t



SCHEMA 3 - Folla compatta (400 kg/mq)

Figure 1.2. Traffic load schemes prescribed by Circular n. 6018 of 9 June 1945 [8].

In 1952 the General Management of ANAS issued Circular n. 820 of 15 March [10], which addressed the design loads on motorways and state roads. The main novelty was the introduction of military-vehicle convoys. The most critical of two combinations had to be chosen for design. The first one used an indefinite train of 61.5-tonne military loads together with one or more parallel columns of 12-tonne trucks and a 400 kg/m² crowd load on the sidewalks. The second one used an indefinite train of 32-tonne military loads with the same accompanying truck columns and crowd load. The introduction of these heavier convoys was a consequence of two factors of that period: the practical needs of military mobility in the early postwar years, and the growing weight of civilian commercial traffic.

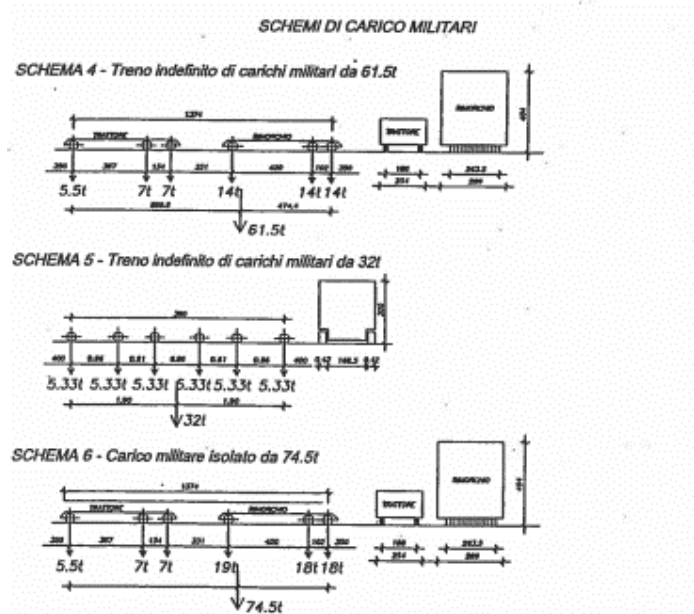


Figure 1.3. Traffic load schemes prescribed by ANAS Circular n. 820 of 15 March 1952 [10].

A wider reorganization of the rules came in 1962 with Ministerial Circular n. 384 of 14 February [11]. The roads were re-classified into two categories: roads open to both civilian and military traffic, and roads open only to civilian traffic. For the first category, three alternative load combinations were prescribed. The engineer had to choose the most onerous one. The three schemes (called Schemes 4, 5 and 6 in the original document) were each combined with one or more 12-tonne truck convoys and with the 400 kg/m² sidewalk crowd load. For the second category, the worst combination was that of one or more 12-tonne truck convoys placed next to one or more steam rollers, with the same crowd load on the sidewalks. The dynamic amplification coefficient was re-formulated as a more refined function of the span:

$$\phi = 1 + (100 - L)^2 / [100 \cdot (250 - L)] \quad (1.2)$$

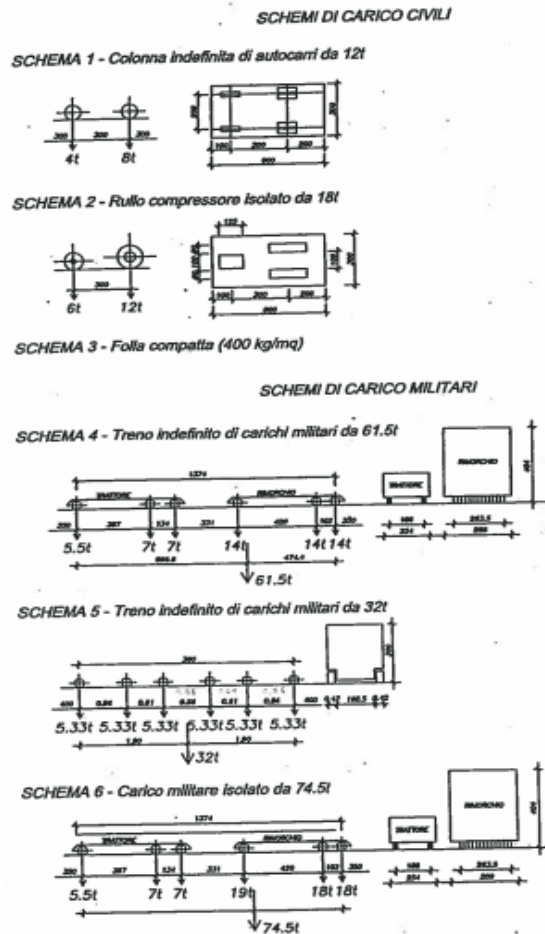


Figure 1.4. Traffic load schemes prescribed by Ministerial Circular n. 384 of 14 February 1962 [11].

A more significant change of philosophy in the Italian bridge code arrived with the Ministerial Decree of 2 August 1980 [12]. This decree listed in detail the loads, weights and deformations to be used in design, and introduced for the first time specific rules for seismic zones. With the 1980 decree, the practice of giving actual vehicle patterns was replaced by equivalent loads, expressed as forces and surface tractions. Three categories of bridges were defined:

- Category I bridges, to be designed for all the moving loads listed in the decree;
- Category II bridges, to be designed only for the loads q1B, q1D, q1E and q1F;
- Category III bridges, limited to q1E and q1F only.

The equivalent loads q1E and q1F corresponded to a 1-tonne concentrated load over a $0.7 \times 0.7 \text{ m}^2$ footprint and to a 0.4 t/m^2 uniformly distributed load, while q1A and q1B were span-dependent functions. For first-category roads, which include motorways and state roads, the design combination required the application of q1A on one lane and of q1B on the next lane, both 3.5 m wide, with the usual 400 kg/m^2 sidewalk crowd load. On any other lane beyond the first two, an additional q1B reduced by 30% was applied. The dynamic amplification

coefficient was updated to depend not only on the loaded length L but also on the ratio between permanent loads g and variable loads q :

$$\varphi = 1.4 - 0.002 \cdot (g/q + 1) \cdot L \quad (1.3)$$

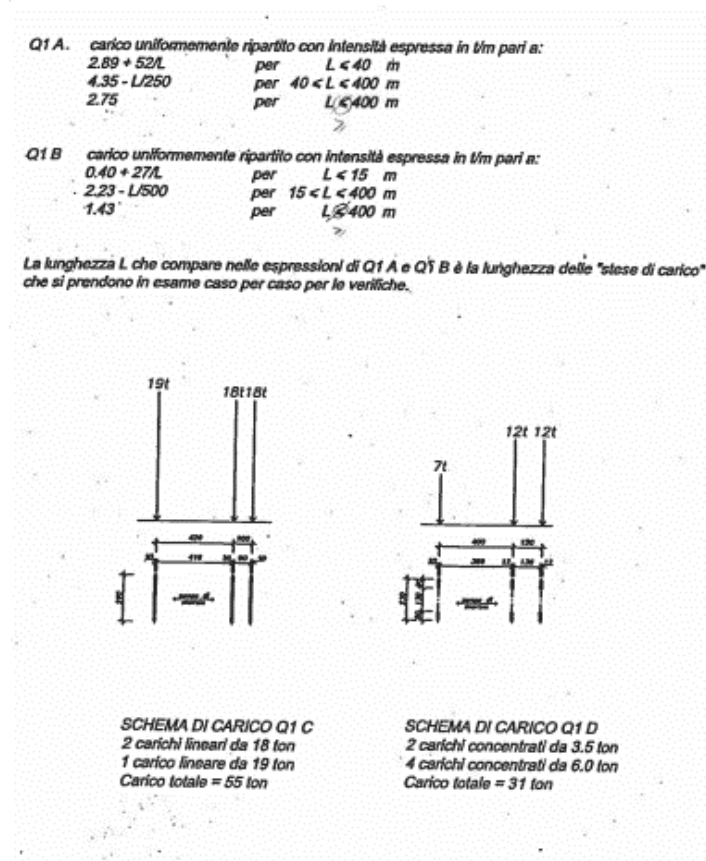


Figure 1.5. Traffic load schemes prescribed by the Ministerial Decree of 2 August 1980 [12].

Ten years later, the Ministerial Decree of 4 May 1990 [13] updated the technical standards for the design, construction and testing of road bridges. This decree is particularly important for the assessment of existing structures, because it introduced for the first time explicit principles for the management of bridges already in service: inspection, monitoring, ordinary and extraordinary maintenance, and static improvement or adaptation. From the design point of view, three categories were kept:

- Category I, for bridges that carry the full set of mobile loads at their full design value;
- Category II, where the $q1A$ and $q1B$ loads are reduced to 75% and 50% of their nominal value;

- Category III, which includes pedestrian walkways, where only the q1E and q1D loads are considered.

For first-category bridges, the combination prescribed by the 1990 decree required the application of q1A (60 tonnes distributed over three axles) together with q1B along the same axis of the lane, on the parts not occupied by q1A. The same scheme was then applied to the adjacent lanes, but with reductions of 50% on the first adjacent lane and 65% on the further ones. The crowd load on the sidewalks remained at 400 kg/m². The dynamic amplification coefficient was again re-formulated, this time as a piecewise function of the span:

$$\varphi = 1.4 \quad \text{for } L \leq 10 \text{ m} \quad (1.4a)$$

$$\varphi = 1.4 - (L - 10)/150 \quad \text{for } 10 \text{ m} < L < 70 \text{ m} \quad (1.4b)$$

$$\varphi = 1.0 \quad \text{for } L \geq 70 \text{ m} \quad (1.4c)$$

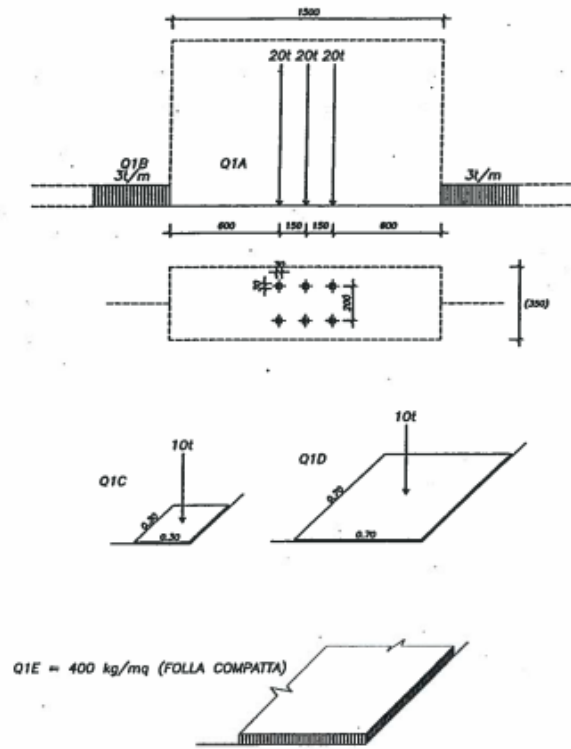


Figure 1.6. Traffic load schemes prescribed by the Ministerial Decree of 4 May 1990 [13].

The current Italian Technical Standards for Constructions (NTC) have their origin in the Ministerial Decree of 14 January 2008 [14], later updated by the Ministerial Decree of 17

January 2018 [15]. Chapter 8 of NTC 2018 is fully dedicated to existing constructions, and it is supported by Circular n. 7 of 21 January 2019 [16], which gives the operational guidance for its application. In the current framework, the load models for road bridges are aligned with those of EN 1991-2 [17]. The dynamic amplification coefficient is no longer applied as an external multiplier, because it is already included in the characteristic values of the load models. Road bridges are now classified according to the type of allowed mobile loads. The standard separates bridges that carry the full set of vehicular load models from pedestrian bridges, for which only Load Model 5 (representing crowds) is used.

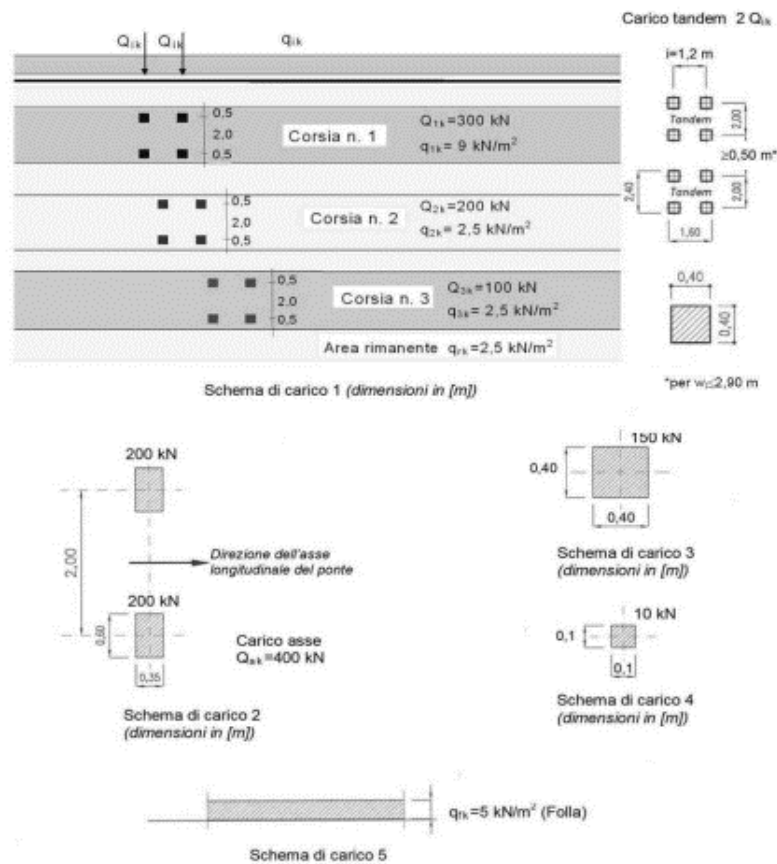


Figure 1.7. Traffic load schemes prescribed by the current NTC 2018 [15] aligned with EN 1991-2 [17].

Table 1.1 summarizes the evolution of the Italian regulatory framework for bridges and shows the main novelty introduced by each standard. Over nearly ninety years, the structure of the documents has slowly shifted from prescribing specific load patterns to defining performance-based criteria, in line with the broader transition toward semi-probabilistic limit-state design.

Table 1.1. Evolution of the Italian regulatory framework for the design of road bridges. The table includes the most relevant standards from 1933 to today.

Standard	Year	Main novelty
Normale n.8	1933	First systematic Italian bridge code; three road categories
Circular 6018	1945	Two-category scheme; dynamic amplification as a function of span
Circular 820 (ANAS)	1952	Introduction of military-load convoys for motorways and state roads
Circular 384	1962	Civilian vs civilian+military distinction; updated ϕ formula
D.M. 02/08/1980	1980	Equivalent loads q1A–q1F replace explicit load patterns; seismic clauses
D.M. 04/05/1990	1990	Modern testing and maintenance principles for existing bridges
D.M. 14/01/2008 (NTC 2008)	2008	Adoption of Eurocode-style load models; ϕ embedded in load values
D.M. 17/01/2018 (NTC 2018)	2018	Current Italian Technical Standards; Chapter 8 on existing structures
Linee Guida Ponti Esistenti	2020 (rev. 2022)	Multi-level approach for the assessment of existing bridges

1.2.2 Guidelines for existing bridges

As mentioned in section 1.1, the collapse of the Morandi Bridge in Genoa on 14 August 2018 had a strong institutional impact. The direct legal consequence was the Decree-Law n. 109 of 28 September 2018 [1], from which the need emerged for a dedicated normative document on the management and safety of the existing Italian infrastructure. Inside this context, the

Guidelines for the classification and management of risk, safety assessment and monitoring of existing bridges [3] were developed and adopted with Ministerial Decree n. 578 of 17 December 2020 [3]. A revised version of the Guidelines was published in 2022, to include the operational experience collected during the first two years of application [4].

The goal of the Guidelines is to obtain a uniform national approach to risk classification, safety management and structural monitoring for existing bridges, viaducts, overpasses and culverts. The document is organized into three main parts, which cover the risk inventory and classification, the inspections, and the monitoring strategy. The most characteristic feature of the Guidelines is the multi-level approach. Rather than prescribing a single uniform procedure for every bridge, which would be impossible given the large number of structures on the national road network, the level of detail of the analysis is scaled according to the attention class assigned to each work. As the level grows, the complexity and cost of the investigation grow as well, but the number of bridges that need this investigation decreases. The six levels of analysis defined by the Guidelines are:

- Level 0, a national census of all bridges and viaducts, with their basic geometric and structural characteristics, based on the available documentation;
- Level 1, visual inspections and surveys of the structures identified at Level 0, with an assessment of their geological and hydraulic context. The aim is to record the observed degradation, identify the risks and verify or refine the geometric and structural data;
- Level 2, evaluation of hazard, vulnerability and exposure for the different sources of risk (structural, seismic, hydrogeological). The result is the assignment of an attention class to each bridge;
- Level 3, a preliminary safety verification, which decides on the basis of the data and damage information gathered at the previous levels whether more refined assessments at Level 4 are necessary;
- Level 4, an in-depth safety assessment of the bridge according to the current technical standards, including, when appropriate, refined numerical analyses;
- Level 5, applied to bridges of strategic importance. At this level, a transportation-network analysis and a socioeconomic impact study are required to evaluate the consequences of a temporary or permanent closure.

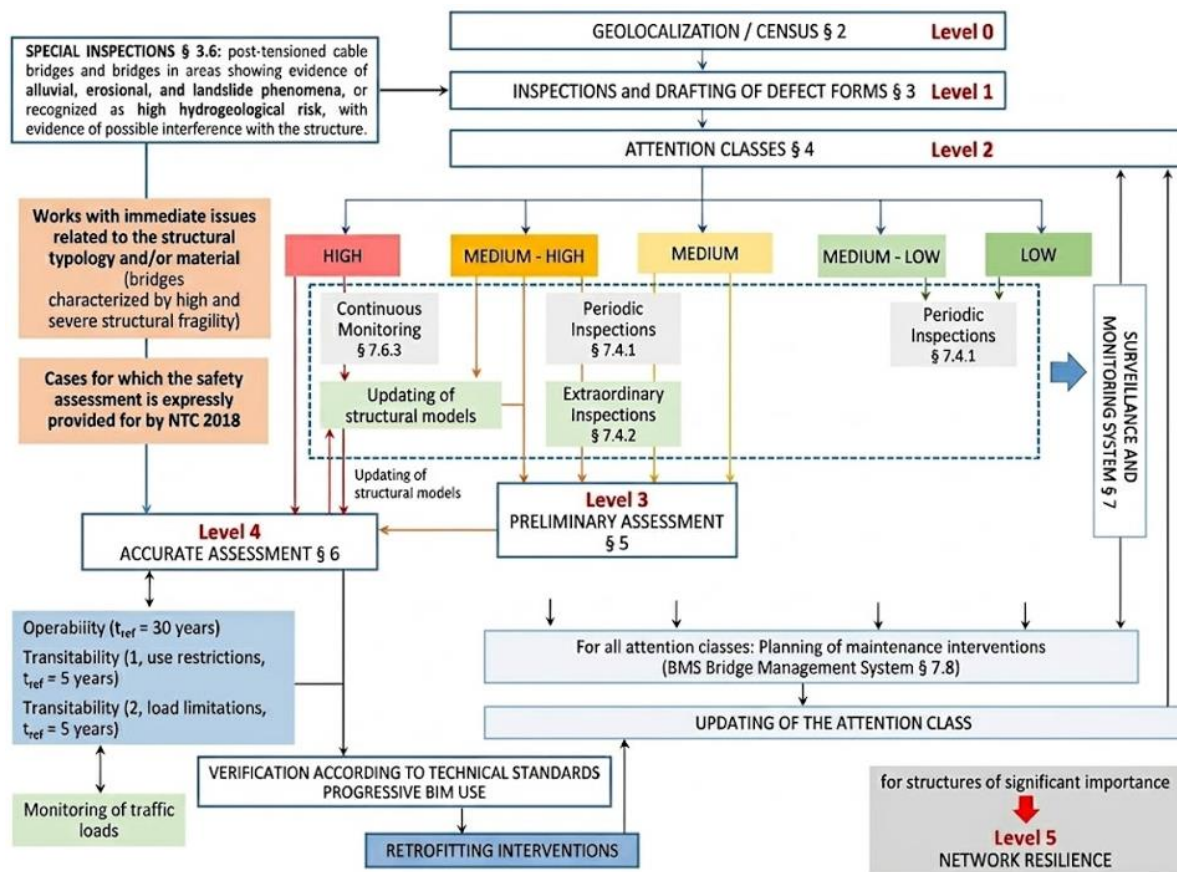


Figure 1.8. Schematic representation of the multi-level approach of the 2020 Italian Guidelines for existing bridges [3].

The central element of this methodology is the attention class. It is assigned to each bridge on the basis of the observed degradation, the structural defects identified, and the static, seismic and hydrogeological risks recognized at the previous levels. Five attention-class levels are defined, from low to high. For bridges in the low or medium-low classes, the Guidelines prescribe ordinary inspections together with a maintenance programme and a periodic update of the class. For the medium and upper-medium classes, periodic inspections are supported by extraordinary inspections whenever rapidly evolving degradation is observed. A preliminary verification corresponding to Level 3 is required to decide if the in-depth Level 4 assessment is necessary. The upper-medium class also allows the introduction of continuous or periodic monitoring, whose results can support the Level 3 decision or trigger a direct passage to Level 4. The highest attention class is associated automatically with the in-depth Level 4 assessment, together with continuous monitoring and periodic inspections.

An important conceptual element of the Guidelines is the explicit statement that knowledge of the historical regulatory framework is essential for the assessment of an existing bridge. The historical loads against which the structure was originally designed must be compared with the

current loads imposed by present-day traffic. This comparison is one of the ingredients of the vulnerability assessment at Level 2 and of the preliminary verification at Level 3, where a simple but informative indicator is the ratio between the internal forces produced by the design loads at the time of construction and those produced by the corresponding modern load models. The detailed historical review presented in Section 1.2.1 is therefore not a digression but a direct prerequisite for the application of the Guidelines.

Inside this framework, the present thesis is a concrete example of application of the Linee Guida at Level 4. The development of a nonlinear finite element model of an actually existing reinforced concrete bridge beam, validated through comparison with a load-to-collapse test, contributes to two distinct aspects of the Guidelines. On the one hand, the in-depth structural verification required at Level 4. On the other hand, the framework of Section 7.5 of the Guidelines themselves, which addresses static load tests and dynamic surveys as means to acquire information on the actual behavior of the structure. The validated numerical model can then be reused as a virtual prototype for the safety assessment of similar bridge girders that belong to the same construction batch or were designed under the same historical standards.

1.2.3 The European framework: Eurocodes and existing structures

The Italian Technical Standards for Constructions are not an isolated regulatory document. Since NTC 2008, and even more clearly with NTC 2018, the Italian rules have been written so that they are compatible with the European framework, which is based on the family of Eurocodes (EN 1990 to EN 1999). The Eurocodes are published by the European Committee for Standardization (CEN) and are formally adopted by each Member State through a National Annex that contains the country-specific values of the so-called Nationally Determined Parameters. For bridges, the most relevant Eurocodes are EN 1990 [24] which, contains the basis of structural design and the rules for the combination of actions; EN 1991-2 [17] which, defines the traffic loads for road, pedestrian and railway bridges; EN 1992-1-1 and EN 1992-2 [25], [26], which give the design rules for concrete structures and for concrete bridges; EN 1993-1-1 and EN 1993-2 for steel bridges; and EN 1998-1 and EN 1998-2 for the seismic design of buildings and bridges respectively. The semi-probabilistic limit-state philosophy, the load factors, and the combination rules of NTC 2018 all come directly from EN 1990 and the related parts of EN 1991.

A key conceptual point is that the current generation of Eurocodes was originally written for new structures. Annex A2 of EN 1990 covers bridges, but in its present form it focuses on the design rather than on the assessment of an existing work. Some specific provisions for existing structures appear in informative annexes of EN 1992 and in the so-called CEN Technical Specifications (CEN/TS), but these are not always adopted by the National Annexes and they vary from one Member State to another. As a consequence, the practical assessment of existing bridges in Europe has been carried out for many years through a combination of the design Eurocodes and the national documents that complement them. In Italy, this means using NTC 2018 together with Chapter 8 of the same document and with the 2020 Linee Guida discussed in the previous section.

This situation is now changing. The so-called "second generation" of Eurocodes, currently under final approval and scheduled for publication between 2026 and 2027, will extend the scope of EN 1990 to the verification of existing structures in an explicit way, and a dedicated part EN 1990:2023 prAnnex E is foreseen for this purpose [27]. A new part of EN 1992 dedicated to the assessment, retrofitting and repair of existing concrete structures (informally referred to as EN 1992-1-1 part 2 or EN 1992-1-3 in the drafts) is also expected to be published shortly. Once adopted, these documents will provide a uniform European framework for the assessment of existing concrete bridges, which complements the multi-level approach of the 2020 Italian Guidelines and supports the use of refined methods such as NLFEA for high-level verifications.

For the case of interest in this thesis, the verification of the 25.90 m beam against the actions defined in the second-generation Eurocodes is not required, because the comparison is made against the experimental load and not against a code-defined design load. However, the choice of the safety formats discussed in Section 1.3 and the use of mean material properties calibrated against destructive tests are coherent with the philosophy that EN 1990 prAnnex E proposes for existing structures, which is based on a transparent treatment of model and parameter uncertainties together with the use of all the information that the engineer can collect on the actual state of the work [27], [28].

1.2.4 The international framework: fib Model Code and JCSS

Above the European level, the assessment of existing concrete structures is also guided by a set of international documents. These documents are not legally binding in Italy, but they provide the technical and probabilistic basis on which the Eurocodes and the NTC are built, and they are often used as a reference when a specific problem is not covered by the national rules. The two most relevant for the present work are the fib Model Code and the documents of the Joint Committee on Structural Safety (JCSS).

The fib Model Code for Concrete Structures, in its 2010 [18] and 2020 [29] editions, is the main international reference for the design and assessment of concrete structures. It is not a code in the legal sense, but a pre-normative document that anticipates the content of future versions of EN 1992. For the assessment of existing structures, the Model Code introduces the levels of approximation mentioned in Section 1.3, and it treats nonlinear finite element analysis as the highest of these levels. It also gives guidance on conservation, on monitoring and on the use of the global resistance formats discussed above. The 2020 edition extends this treatment and pays more attention to existing structures and to their management through the service life.

The Probabilistic Model Code of the JCSS [30] complements the Model Code on the side of reliability. It collects recommended probability distributions for material properties, geometrical quantities, loads and model uncertainties, together with target reliability values. These distributions are the natural input for the probabilistic and semi-probabilistic safety formats of Section 1.3, and they are used, for example, to justify the coefficient of variation of the global resistance in the ECoV method. Two ISO standards complete the picture. ISO 2394 [31] states the general principles on reliability for structures and links the target reliability to the consequences of failure and to the cost of the safety measures. ISO 13822 [32] is dedicated to existing structures, and it allows the engineer to update the prior assumptions with the information collected through inspection and testing. The mean material properties used in this thesis, obtained from destructive tests on the beam, are an example of this kind of updated information, and their use is consistent with the philosophy of ISO 13822 and of the JCSS code [30], [32].

1.3 Safety formats for nonlinear finite element analysis

The use of finite element models that explicitly include material nonlinearity, and where relevant geometric nonlinearity, changes the way the safety verification is performed in a fundamental way. In the conventional linear-elastic approach, the structure is analyzed assuming linear material behavior, the internal actions are obtained from the global model, and the verification is then carried out section by section by comparing the local design action E_d with the local design resistance R_d :

$$E_d \leq R_d \quad (1.5)$$

This local approach has been the cornerstone of structural engineering practice for several decades. It is intrinsically conservative, because each section is verified independently from the others, and the redistribution of internal forces that the structure can sustain in the post-elastic range is not exploited. The nonlinear analysis, on the contrary, allows the engineer to take advantage of the plastic resources of the materials and of the effective stiffness redistribution. The result is a more realistic, and usually larger, estimate of the global resistance of the structure. So the verification itself must be re-formulated in global terms. The global design action F_d is compared directly with the global design resistance R_d obtained from the nonlinear analysis:

$$F_d \leq R_d \quad (1.6)$$

The fib Model Code 2010 [18] classifies the available methods for the assessment of concrete structures according to four levels of approximation (LoA I to LoA IV). A higher level corresponds to a more accurate but also more demanding analysis. NLFEA falls within the highest level of approximation. As discussed in detail in the literature [19], [20], it requires the use of a global resistance format (GRF). Within this format, the verification condition is expressed as:

$$R_d = R_{rep} / (\gamma_R \cdot \gamma_{Rd}) \quad (1.7)$$

where R_{rep} is the representative resistance obtained from the nonlinear analysis. Its precise definition depends on the specific safety format used. The coefficient γ_R is a global partial factor that accounts for the random (aleatory) uncertainties associated with material strengths and geometrical dimensions, while γ_{Rd} is a separate partial factor that accounts for the epistemic uncertainties associated with the numerical model itself (constitutive law, mesh, boundary

conditions, solver). Equation (1.7) is therefore consistent with the conventional semi-probabilistic philosophy of the limit-state approach, but it shifts the application of the partial factors from the local section-level resistance to the global structural resistance.

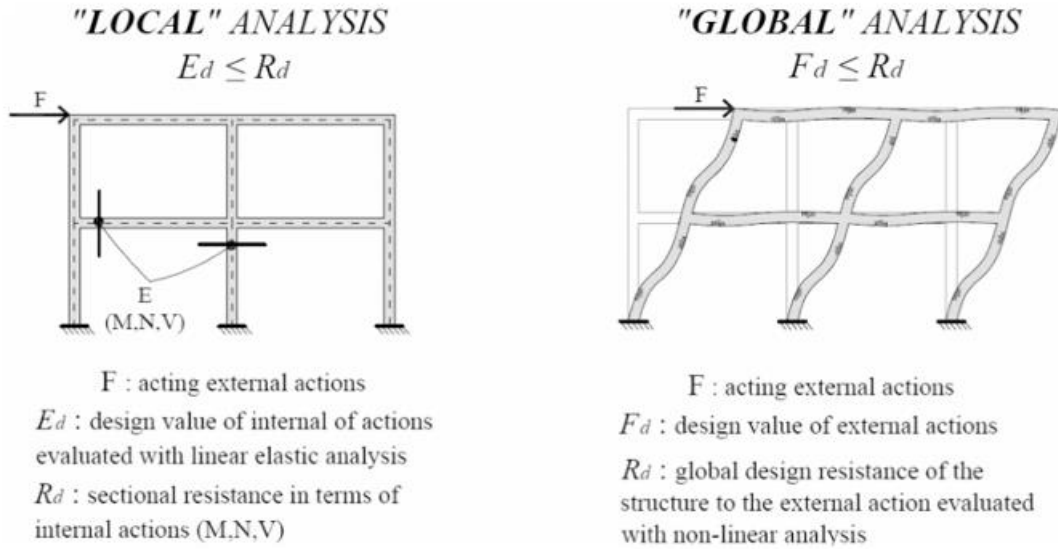


Figure 1.9. Conceptual difference between the local approach (left) typical of linear-elastic analysis and the global approach (right) typical of NLFEA. Adapted from Allaix et al [19].

In real practice, the global design action is rarely produced by a single load. Usually a combination of permanent, variable and accidental actions must be considered. Inside an NLFEA performed in displacement or load control, the load combination is applied through a load multiplier λ that grows monotonically until the model reaches its global capacity. The global design resistance is then expressed as [20]:

$$R_d = (\lambda_{u, NLFEA} \cdot F_d) / (\gamma_R \cdot \gamma_{Rd}) \quad (1.8)$$

where $\lambda_{u, NLFEA}$ is the ultimate load multiplier obtained from the nonlinear analysis, while F_d is the reference design load combination. For example, the fundamental combination prescribed by NTC 2018 [15] reads:

$$F_d = \gamma_{G1} \cdot G_1 + \gamma_{G2} \cdot G_2 + \gamma_P \cdot P + \gamma_{Q1} \cdot Q_{k1} + \psi_{02} \cdot \gamma_{Q2} \cdot Q_{k2} + \dots \quad (1.9)$$

Several safety formats compatible with NLFEA have been proposed in the literature and included in the fib Model Code 2010. The most widely discussed are the Partial Factor Method (PFM), the Estimate of the Coefficient of Variation method (ECoV), the Global Resistance Factor method (GRFM), the Global Safety Format (GSF) in its most general form, and the fully Probabilistic Method (PM). The next paragraphs review each of these formats. The aim is to

describe their conceptual basis and the trade-off they offer between computational cost and accuracy. The discussion is focused on the conceptual structure of each format; complete derivations and operational guidance can be found in the references cited.

- **Partial Factor Method (PFM).**

The Partial Factor Method [20] is the most direct extension of the conventional partial-factor approach to NLFEA. The nonlinear analysis is carried out using the design values of the material strengths f_d (the characteristic values already divided by the corresponding material partial factors) and the nominal geometric quantities a_d . The global design resistance is then obtained by dividing the resulting global resistance by the model-uncertainty factor γ_{Rd} :

$$R_d = R_{NLFEA}(f_d; a_d) / \gamma_{Rd} \quad (1.10)$$

The PFM has the advantage of needing only a single nonlinear analysis, and of being conceptually consistent with the traditional design philosophy. Its main drawback is that the use of strongly reduced design material strengths inside a nonlinear analysis can change the predicted failure mechanism. The strong reduction of concrete tensile and compressive resistances may push the model to develop a different crack pattern or a different sequence of local failures from what would happen in reality. This effect is particularly visible for shear-dominated or shear-flexure failure modes.

- **Estimate of the Coefficient of Variation (ECoV) method.**

The ECoV method, proposed by Červenka [21] and adopted by the fib Model Code 2010, addresses the limitation of the PFM. The nonlinear analysis is performed using the mean values of the material strengths f_m and the nominal geometric values a_n . The global resistance computed under these assumptions, R_m , is taken as the central estimate of the global capacity. A second nonlinear analysis is then performed using the characteristic values f_k , which gives a lower estimate R_k . Under the assumption of a log-normal distribution for the global resistance, the coefficient of variation due to material uncertainties is estimated as:

$$V_{Rm} = (1 / 1.65) \cdot \ln(R_m / R_k) \quad (1.11)$$

and the global random-uncertainty partial factor is computed as:

$$\gamma_R = \exp(\alpha_R \cdot \beta \cdot V_R) / \delta_R \quad (1.12)$$

$$V_R = \sqrt{(V_{Rm}^2 + V_{Rg}^2)} \leq 0.30 \quad (1.13)$$

In the equations above, V_{Rg} is the coefficient of variation of the global resistance associated with geometric uncertainties, usually taken equal to 0.05 for non-slender elements. The parameter α_R is the sensitivity factor (equal to 0.8 under the assumption that the random material and geometric uncertainties dominate over the epistemic ones), and β is the reliability index associated with the desired probability of failure, which depends on the target reliability for the limit state under consideration. The bias factor δ_R combines the contributions of the geometric and material biases:

$$\delta_R = \delta_{Rm} \cdot \delta_{Rg} \quad (1.14)$$

$$\delta_{Rm} = \mu_{Rm} / R_{NLFEA}(f_m; a_n) \quad (1.15)$$

For non-slender members, δ_{Rg} and δ_{Rm} are normally taken equal to one. The main advantage of the ECoV method is its computational efficiency. Only two nonlinear analyses are required to obtain a probabilistically informed estimate of the global resistance, which makes the method attractive for routine engineering applications where a full probabilistic analysis is not feasible.

- **Global Safety Format (GSF).**

The Global Safety Format adopts the same conceptual framework as the ECoV method, but replaces the closed-form estimate of V_{Rm} with its direct numerical evaluation through a probabilistic simulation [19]. Multiple nonlinear analyses are performed, each one with a different realization of the material properties sampled with a Monte Carlo or a Latin Hypercube scheme. The standard deviation of the resulting set of global resistances is then divided by the mean to obtain V_{Rm} :

$$V_{Rm} = \sigma_R / \mu_R \quad (1.16)$$

The GSF is more accurate than the ECoV but requires a significantly larger computational effort, usually of the order of 30 to 100 nonlinear analyses, depending on the convergence of the sampling. It is therefore more suitable for research applications or for structures of particular strategic importance, where a full quantification of the resistance distribution is justified.

- **Global Resistance Factor (GRFM) method.**

The Global Resistance Factor method, included in the fib Model Code 2010 [18] and in EN 1992-2 [26], offers an even simpler approach. A single nonlinear analysis is performed using modified material values defined as $1.1 \cdot f_{yk}$ for the reinforcing steel and $0.85 \cdot f_{ck}$ for the concrete. The resulting global resistance is then divided by a single global resistance factor γ_{GL} set equal to 1.27:

$$R_d = R_{NLFEA}(1.1 \cdot f_{yk}, 0.85 \cdot f_{ck}) / \gamma_{GL} \quad (1.17)$$

The choice of the multipliers 1.1 and 0.85 reflects an attempt to balance the underestimation introduced by the use of characteristic values with the need to maintain a realistic failure mechanism. The fixed value $\gamma_{GL} = 1.27$ was calibrated against numerical and experimental benchmarks across a wide range of structural typologies. The GRFM method is computationally as efficient as the PFM and therefore competitive for routine applications, although its constant safety factor cannot adapt to the specific uncertainty of the structure being analyzed.

- **Probabilistic Method (PM).**

On the opposite side of the spectrum, the fully Probabilistic Method considers a complete probabilistic description of the material strengths and geometrical quantities. The design resistance is obtained by reading the value associated with a specific quantile of the resulting distribution of the global resistance:

$$R_d = R(\alpha_R, \beta) / \gamma_{Rd} \quad (1.18)$$

where the quantile is identified by the product of the sensitivity factor α_R and the reliability index β . The reliability index β is in turn related to the target probability of failure P_f through the inverse of the standard normal cumulative distribution function:

$$\beta = -\Phi^{-1}(P_f) \quad (1.19)$$

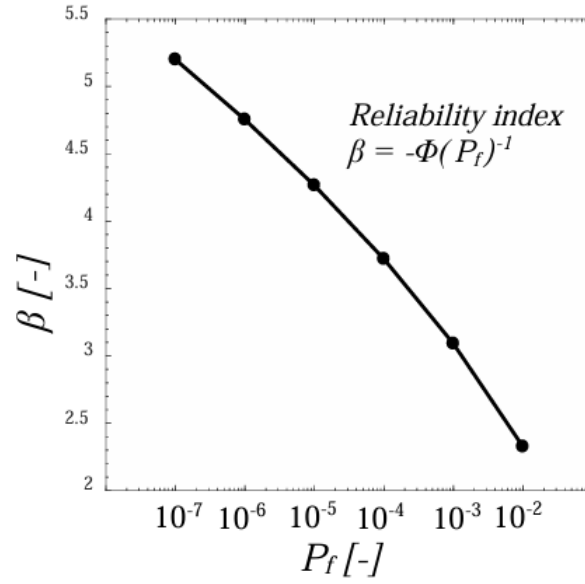


Figure 1.10. Relationship between the reliability index β and the probability of failure P_f , adapted from Šýkora et al. [23].

As discussed in [20] the main advantage of the fully Probabilistic Method is its ability to capture the variability of the failure mechanism itself. By sampling the entire space of material realizations, the method can detect cases in which different combinations of strengths lead to different failure modes. This is not possible with deterministic formats that fix a single set of strengths. The cost of this generality is a very high computational effort, which limits the practical application of the method to a relatively narrow set of cases.

In the present thesis, the nonlinear finite element model is calibrated against experimental data, with the explicit goal of reproducing the load-displacement and strain response measured during the load-to-collapse test on the 25.90 m beam. For this reason, the analysis is performed using mean values of the material properties, obtained from the destructive tests on concrete cores and rebar samples extracted after the test and reported in Chapter 2. This choice is consistent with the central estimate of the global resistance that is used as input for the ECoV, GSF and PM safety formats, and it is the most appropriate reference for the model validation. The model produced in this way is suitable for the subsequent application of any of the safety formats described above, depending on the level of accuracy required by the specific assessment problem.

1.4 Sources of uncertainty in nonlinear finite element analysis

Every structural analysis carries uncertainty, and a nonlinear finite element analysis is no exception. The reliability-based safety formats of Section 1.3 are meaningful only if the uncertainties that affect the result are identified and, as far as possible, quantified. It is common to separate them into two broad groups. The first group is the aleatory uncertainty, which is the natural and irreducible scatter of physical quantities. The second group is the epistemic uncertainty, which comes from the limited knowledge of the analyst and can in principle be reduced by collecting more information or by using a better model [30], [31].

In a nonlinear model of a reinforced concrete member, the aleatory uncertainty is mostly carried by the material properties and by the geometry. The compressive and tensile strength of concrete, its modulus of elasticity and its fracture energy all show a significant scatter from one point of the structure to another, and this scatter is larger in an existing structure that has aged and degraded. The properties of the reinforcing steel, although more stable, also vary around their mean values. Geometric uncertainty includes the real dimensions of the cross-section, the position of the bars and the thickness of the concrete cover, which can differ from the design drawings because of the construction tolerances and of the loss of cover [28], [33].

The epistemic part is dominated by the so-called model uncertainty. A numerical model is always a simplification of reality. The constitutive laws, the type of finite element, the description of the bond between steel and concrete and the boundary conditions are all approximations, and each of them introduces an error between the computed response and the real one. This error is described by a model uncertainty factor, defined as the ratio between an experimental resistance and the resistance predicted by the model under the same conditions:

$$\theta = R_{exp} / R_{NLFEA} \quad (1.20)$$

A value of the factor larger than one means that the model underestimates the real resistance, while a value smaller than one means that it overestimates it. The statistical treatment of this factor over a large set of benchmark tests gives the mean and the coefficient of variation that feed the model partial factor γ_{Rd} of Equation (1.7). Castaldo, Gino and Mancini [34] and

Holicky and Sykora [33] discuss this factor in detail for reinforced concrete and propose values for use in the global resistance formats.

A further source of error is purely numerical. It includes the discretization error linked to the size and the type of the mesh, the error of the iterative solver and the tolerance adopted for convergence. Unlike the physical uncertainties, this numerical error is reducible. It can be controlled by refining the mesh, by checking the convergence of the solution and by following the verification and validation procedures that are now standard in computational mechanics [35], [36], [37], [38]. For existing structures, the reduction of the epistemic uncertainty is particularly valuable, because the information gathered through inspection and testing, as required by the Guidelines of Section 1.2.2, lowers the uncertainty on geometry and materials and therefore narrows the partial factors.

Chapter 2. Load test on the reinforced concrete beam

2.1 Introduction

The experimental reference for this thesis is a load-to-collapse test on a reinforced concrete bridge beam of 25.90 m span. The beam is one of two girders removed from a decommissioned viaduct on the southbound carriageway of the A6 motorway, and it was moved to a dedicated test site where it could be loaded to failure under controlled conditions. The second girder, a 34.60 m prestressed concrete beam, falls outside the scope of the present work. The test took place in July 2021. The load was applied by stacking cast iron ballast on the slab in successive layers, until the beam could no longer carry it. Every experimental quantity used in this chapter, that is the geometry, the material properties, the description of the test and the recorded measurements, comes from the technical report of the Department of Structural, Geotechnical and Building Engineering (DISEG) of the Politecnico di Torino [6].

The beam was an edge element of the original deck, and its cross-section is therefore strongly asymmetric about the vertical axis. This asymmetry has a direct mechanical consequence. Because the centroid and the principal axes of an asymmetric section are offset from the vertical, a purely vertical load no longer produces simple bending in the vertical plane. It generates instead a combination of bending about an inclined axis and a torsional moment. The beam therefore responds in skew bending even under vertical loading alone, and this behavior shaped both the preparation of the test and the mechanism by which it eventually failed. A number of the operations carried out before and during the test were intended specifically to reduce the skew bending and the torsion that accompany it, as described in Section 2.5, and the role of the asymmetry recurs throughout the sections that follow.

The measurements collected during the test form the reference set against which the nonlinear finite element model of Chapter 3 is first calibrated and then validated. The description given here therefore concentrates on the quantities that the model must be able to reproduce, in particular the relationship between the total applied load and the vertical displacement at the midspan, and the strain developed in the main tensile reinforcement.

2.2 Test field

Once they had been removed from the viaduct, the two beams were transported, in agreement with SINA S.p.A., to a former motorway service area close to the A6 motorway. A dedicated test field was prepared inside this area to carry out the load-to-failure tests on both beams [6]. The site had to accommodate two activities at the same time, the work of the technical staff responsible for the monitoring and the work of the operators handling the heavy loads, and it was sized so that both could proceed safely. It offered vehicle access, enough room for the cranes used in the lifting operations, space for the storage of materials and equipment, and room to build the auxiliary structures required by the test.

For the test on the 25.90 m beam, the area around the girder was equipped with two cranes and an operator basket for the work carried out above the beam. Two zones were set aside, one for the hydraulic jacks and one for the eighteen ballast units of 105 kN that were later placed on the structure. A temporary roofed shelter housed the technical staff, the power generator and the data acquisition equipment [6].

The beam was simply supported on two reinforced concrete foundation blocks, referred to here as plinth A and plinth B. Each plinth carries a vertical wall against which the Gerber half-joint at the end of the beam bears, so that the beam transfers its support reactions to the plinths through these half-joints. On one side of the supports a lateral abutment was built, because the asymmetry of the edge beam leaves half of the end crossbeam missing on that side. The bearings themselves are neoprene sheets 3 cm thick. A trench was excavated across the midspan section to house the jacks, the timber supports, the steel plates and the sensors that together form the central support system described in Section 2.5.

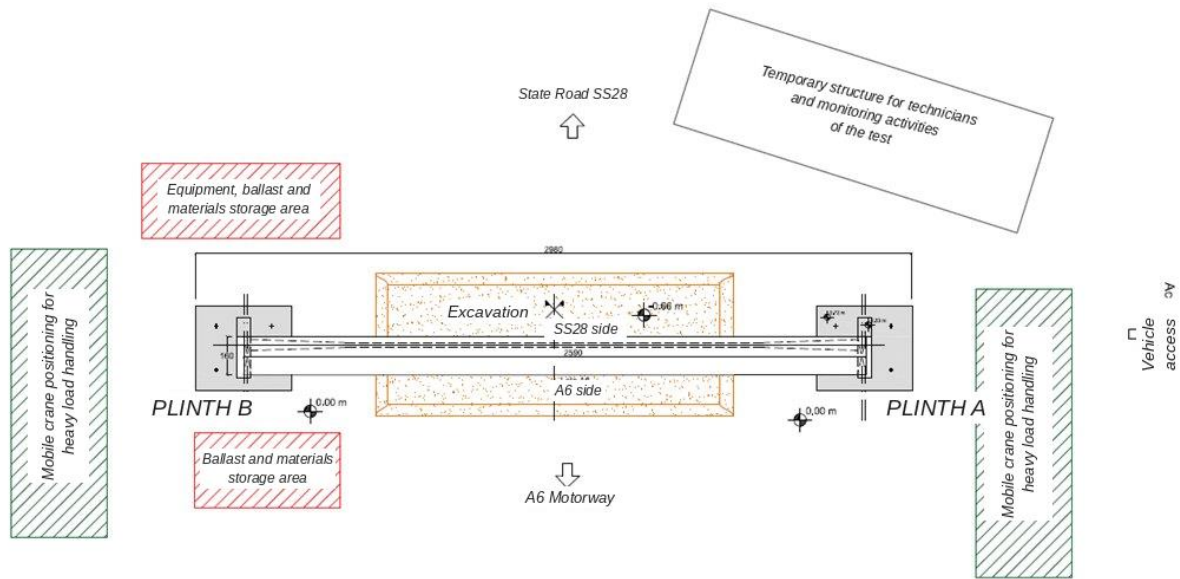


Figure 2.1. Layout of the test field at the former motorway service area.

2.3 Geometry and reinforcement

2.3.1 Cross-section

The element chosen for the test is an edge beam of the original deck. Its geometry was reconstructed from the original design documents, namely the static calculation report of the viaduct and the reinforcement drawings, and it was then verified against a survey carried out on the beam itself [6]. The total height of the section is 1.85 m. The cross-section is asymmetric and is made of two parts cast at different times. The lower part is a precast reinforced concrete main beam, while the upper part is a cast-in-place reinforced concrete slab, 20 cm thick, that extends mainly towards one side. This lateral extension of the slab is the source of the asymmetry, since the beam originally occupied the edge of the deck.

When the deck was demolished, a longitudinal cut separated the beam from the rest of the structure. Portions of the cast-in-place crossbeams remained attached to the element after this cut, at the two supports and at the midspan. The clear span between the supports is 25.90 m.

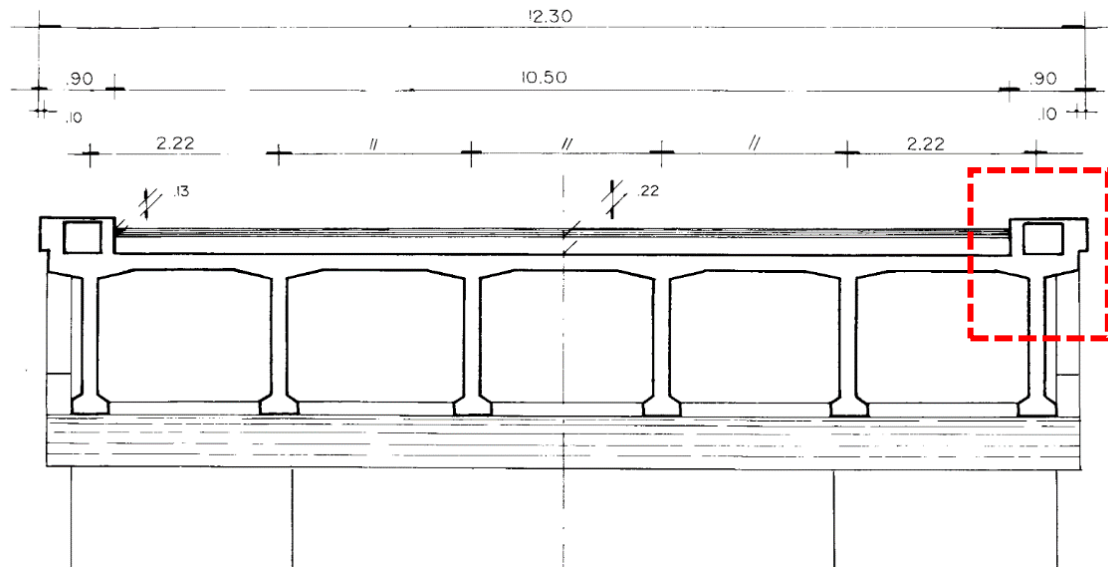


Figure 2.2. Cross-section of the original deck with the tested edge beam highlighted.

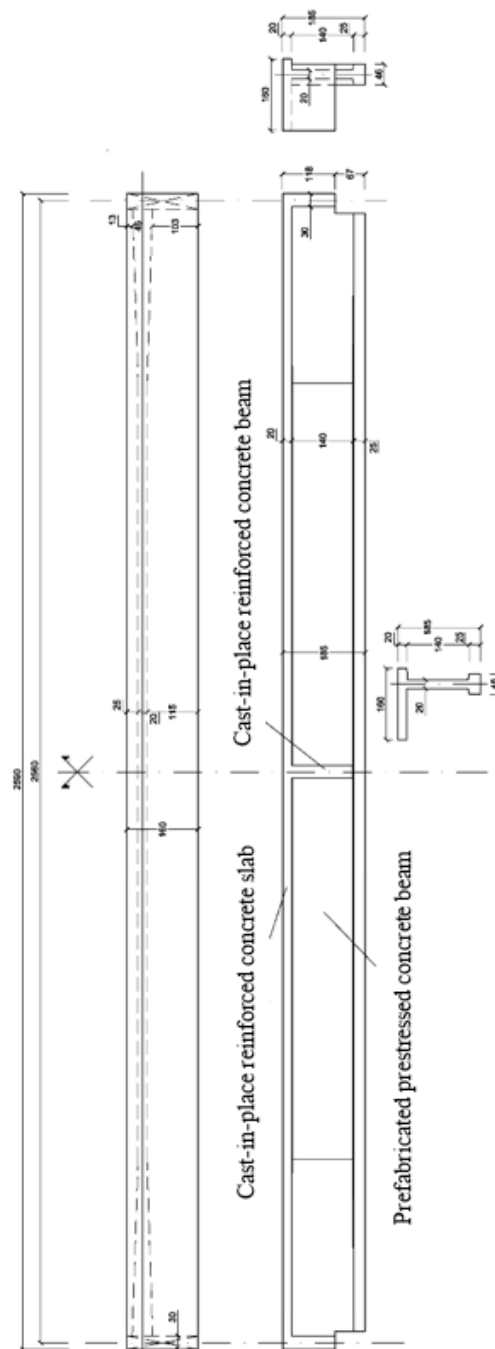


Figure 2.3. Geometry of the 25.90 m beam.

2.3.2 Ordinary reinforcement

The principal reinforcement lies in the lower chord and consists of twenty bars of 28 mm nominal diameter ($20\Phi28$). These bars are bent upward towards the ends of the beam, so that near the supports they also contribute to the shear resistance. The web is reinforced with stirrups

of 8 mm diameter ($\Phi 8$). The slab carries transverse bars of 8 mm diameter ($\Phi 8$) at a spacing of 20 cm, together with longitudinal bars of 12 mm diameter ($\Phi 12$). The complete layout of these bars is defined in the original design drawings [6].

All the bars are of the ribbed type and have a square cross-section, a detail later confirmed by the samples taken for the laboratory tests (Section 2.8). The numerical model of Chapter 3 reproduces this reinforcement in full. It includes the $20\Phi 28$ bars of the lower chord, the $\Phi 8$ stirrups of the web, and the $\Phi 8$ transverse bars and the $\Phi 12$ longitudinal bars of the slab.

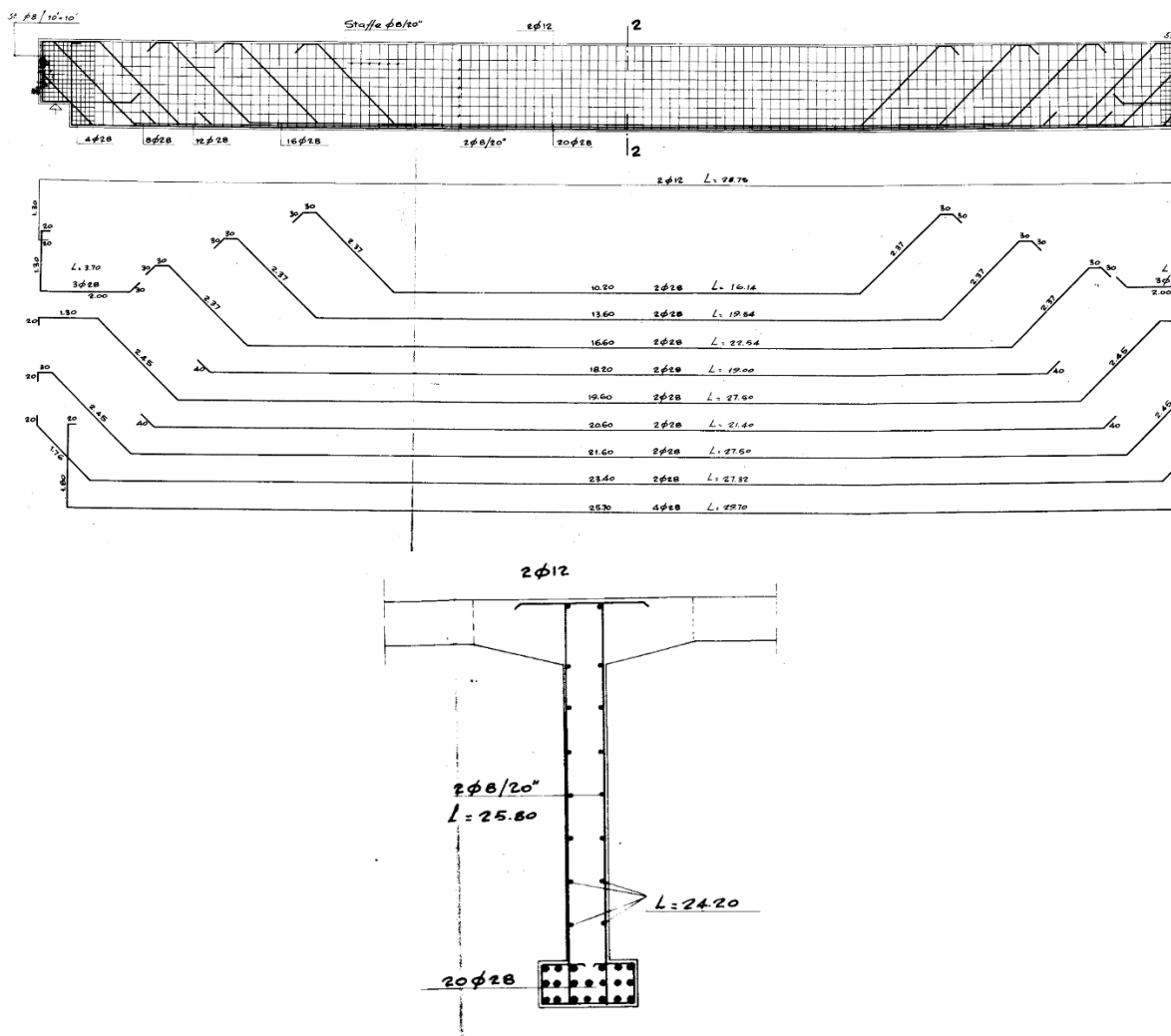


Figure 2.4. Reinforcement layout of the beam and of the slab.

2.4 Materials and degradation

2.4.1 Design materials

Before the destructive tests of Section 2.8 became available, the material properties were taken from the historical documentation of the viaduct. The design strengths of the concrete and the catalogue values of the steel were used both to plan the test and to estimate the resistance of the beam in advance. Where a property was not reported in the original documents, it was derived from the available values using the relations of the current standards [6] [25].

Two classes of concrete were specified for the deck. The cast-in-place concrete of the slab and of the crossbeams belongs to class 250, with a characteristic cubic strength R_{ck} of 25 MPa, whereas the precast concrete of the main beam belongs to class 350, with a characteristic cubic strength R_{ck} of 35 MPa. The corresponding cylindrical strengths and elastic moduli, obtained through the expressions of EN 1992-1-1 [25] are listed in Table 2.1. The ordinary reinforcing steel was specified as grade AQ-42, equivalent to FeB44k with improved bond. Its characteristic yield strength is at least 440 MPa, its mean yield strength is 484 MPa and its design yield strength is 383 MPa, with an elastic modulus of 200 GPa.

Table 2.1 .Design properties of the concrete assumed from the original documentation, for the cast-in-place slab and crossbeams (class 250) and for the precast beam (class 350).

Property	Cast-in-place (class 250)	Precast (class 350)
R_{ck} [MPa]	25	35
f_{ck} [MPa]	20.75	29.05
f_{cm} [MPa]	28.75	37.05
f_{cd} [MPa]	13.83	19.36
f_{ctm} [MPa]	2.26	2.84
E_c [MPa]	30200	32558

2.4.2 Observed degradation

The beam had stood in service for more than sixty years as an edge element of the deck, exposed to the environment on several of its faces. The inspection performed before the test, together with the observations made after the collapse, allowed the principal forms of degradation to be identified [6].

The most extensive form of damage is the loss of the concrete cover along the lower chord. It originates from the corrosion of the $20\Phi 28$ main bars and appears at several positions along the beam and on both of its sides. The same combination of bar corrosion and cover loss, affecting both the main bars and the web reinforcement, was found at the Gerber half-joint of support A. A further sign of deterioration is a longitudinal crack along the intrados of the slab, produced by the corrosion of the $\Phi 8$ transverse bars.

An earlier strengthening intervention had covered the entire beam with a layer of fiber reinforced cementitious mortar about 3 cm thick. This layer is poorly bonded to the concrete beneath it and is prone to detachment. It also hid the surface of the element, so that any local damage accumulated over the decades of service could not be detected before the test.

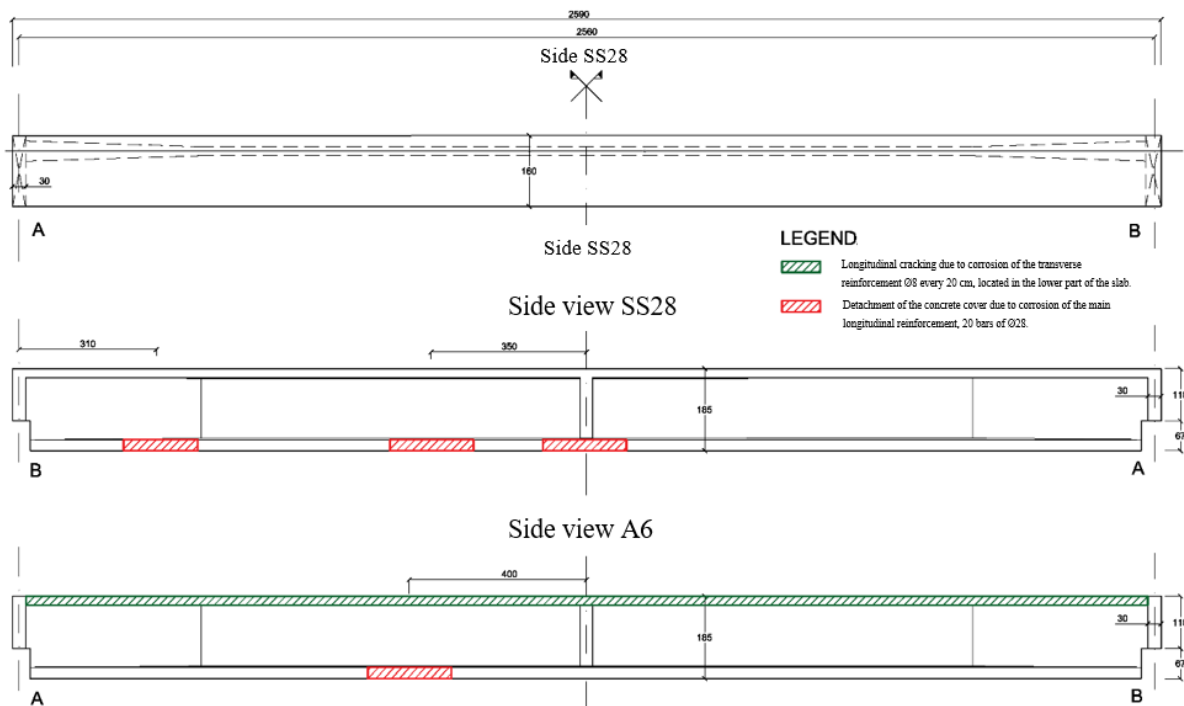




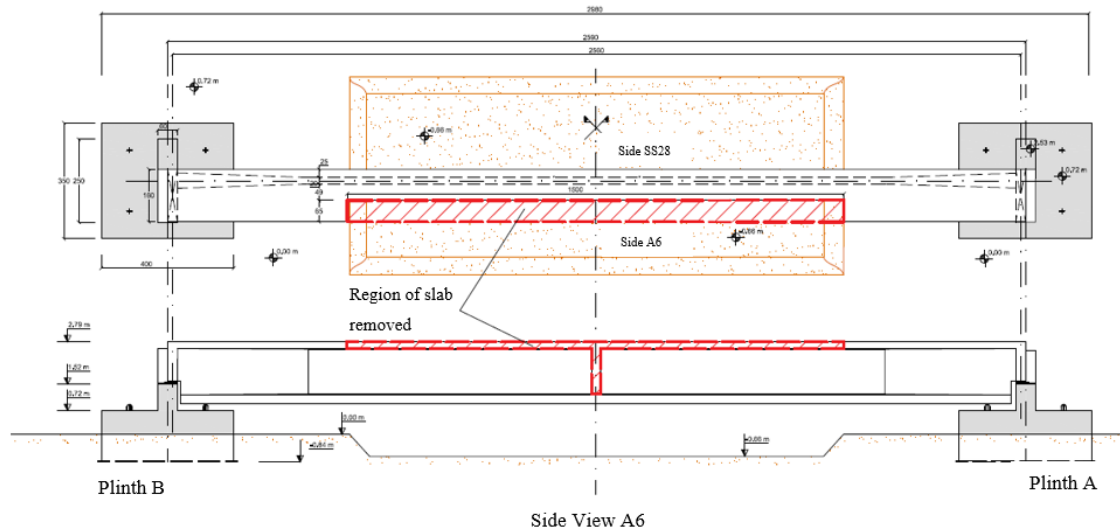
Figure 2.5. Main types of degradation observed on the beam.

2.5 Load test execution

2.5.1 Test setup

The load was applied by placing cast iron ballast units directly on the slab. Each unit weighs 105 kN, about 10.5 t. The units were arranged symmetrically with respect to the midspan over a length of roughly 12 m, so that the loaded region reproduces, in an approximate way, both the footprint and the weight of a heavy vehicle crossing the structure [6].

The asymmetry of the cross-section was the main difficulty to be addressed in setting up the test, because it makes the beam bend out of the vertical plane and twist even under vertical loads. Two measures were adopted to limit these effects. The first was the removal of a strip of the slab in the central region of the beam, 15 m long and 0.65 m wide. This cut was made on site a few days before the test. Over most of the length it leaves a section that is more nearly symmetric about the vertical axis, which favors bending in the vertical plane under the applied load. The second measure was the construction of two steel restraint devices, placed at the lower jacks and arranged symmetrically about the midspan, at a distance of 2.75 m from it on either side, in order to restrain the torsional rotation of the beam.



2.5.2 Loading procedure

To develop the largest possible bending moment at the midspan, and to keep the load as concentrated as possible, the ballast was placed along a central band 11.06 m long, centered on the midspan section. From the geometry and the mean material properties available at that stage, the bending resistance of the beam was estimated at 10728 kN·m [6].

The loading plan was based on four layers of ballast, denoted SA, SB, SC and SD. Each layer is made of four units and therefore weighs 420 kN, so that the four layers together would have applied a total of 1680 kN. This target was chosen to reproduce, in qualitative terms, the effect of four heavy vehicles of 44 t each. The load was not applied as a single monotonic ramp. It followed a sequence of loading and unloading cycles, which made it possible to observe the progressive loss of stiffness of the beam at increasing load levels. In what follows, a loading step denotes the placement or the removal of a single ballast unit, and a loading cycle denotes a complete sequence of loading followed by unloading.

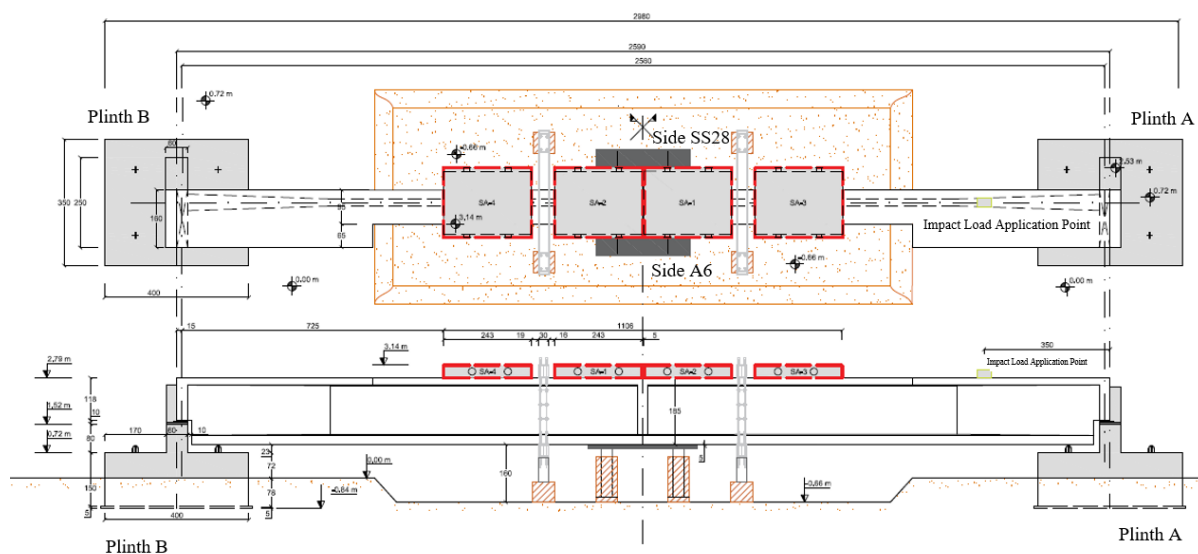
The test was completed in a single day, in July 2021. The two preceding days were used to prepare the field, to survey the geometry and the existing damage and to install the monitoring devices, while the following days were used to survey the damage, to extract the material samples described in Section 2.8 and to recover the instruments. Table 2.2 summarizes the load history by cycle, giving the ballast layers in place and the peak load reached in each cycle, and the complete sequence of loading steps is shown in Figure 2.8.

At the end of the second cycle, after loading step 16, the beam developed a torsional rotation towards the side of the state road SS28. To limit this rotation, timber shims were driven in near the supports, in the gap between the vertical wall of the plinths and the face of the beam towards the SS28. During the pause of about two hours that followed, the position of the ballast was also corrected with greater care, with the aim of reducing the torsion and the skew bending and of promoting bending in the vertical plane. As shown in Section 2.7, these two operations reduced the horizontal displacements during the later stages of the test. The lateral restraint provided by the shims is represented in the numerical model of Chapter 3.

The loading sequence continued until collapse. Failure occurred during the placement of the third layer, at loading step 44, for a total ballast load of 1260 kN, that is 126 t. The beam therefore failed below the planned maximum of 1680 kN. The failure mechanism and the response measured up to that point are described in Section 2.7.

Table 2.2. Load history of the test summarized by loading cycle, with the ballast layers, the loading steps and the peak total load of each cycle.

Cycle	Ballast layers	Loading steps	Unloading steps	Peak load [kN]
1	SA	1 to 4	5 to 8	420
2	SA	9 to 12	13 to 16	420
3	SA + SB	17 to 24	25 to 32	840
4 (to collapse)	SA + SB + SC	33 to 44	-	1260



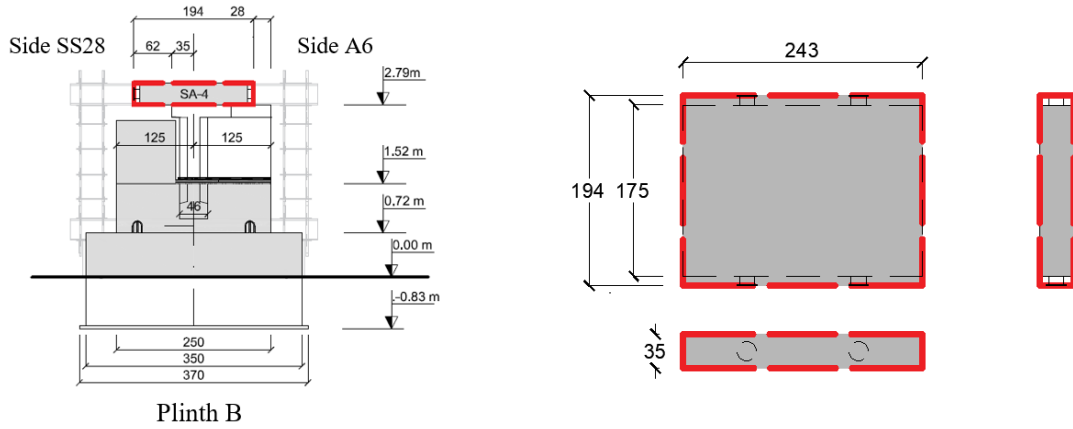


Figure 2.7. Ballast unit and arrangement of the loading layers.

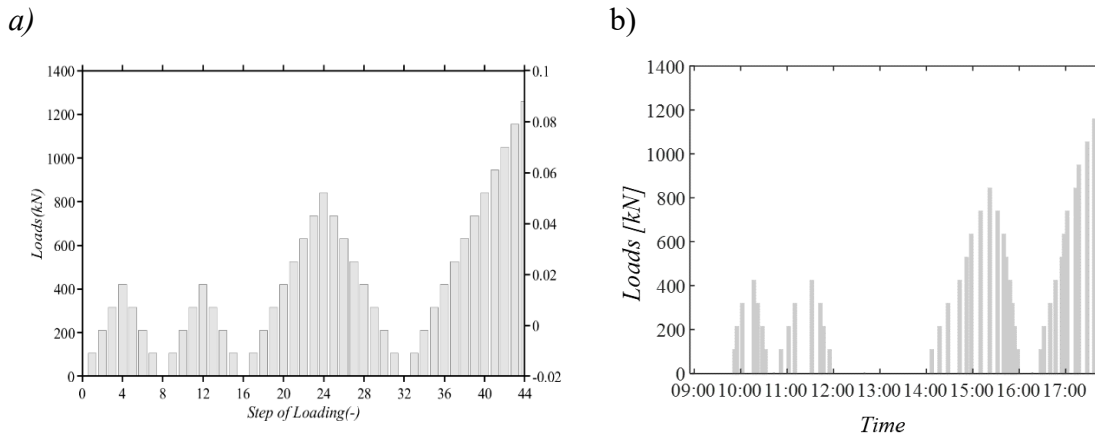


Figure 2.8. Load history, as total load against a) loading step and b) against time.

2.6 Monitoring systems

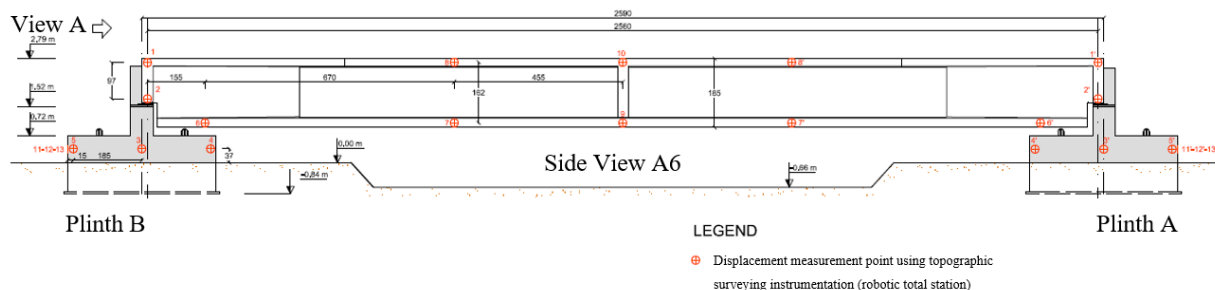
Three monitoring systems are used in this thesis to follow the response of the beam during the test: a network of topographic targets, a set of LVDT extensometers and two strain gauges applied to the main reinforcement. The three systems are described in this section, and the measurements they provided are reported in Section 2.7.

2.6.1 Topographic targets

The displacements of the beam were measured by a topographic network. Four robotic total stations took readings in turn, at regular time intervals, on a set of 24 numbered targets fixed to the beam and to the two plinths [6]. The targets are numbered from 1 to 13 and from 1' to 13' except for 9' and 10'. The primed labels identify the targets on the plinth B side and at the corresponding symmetric positions along the beam, while the midspan section carries two targets (9 and 10) that have no primed counterpart. In the notation used here, each target is written as the letter P followed by its number, for example P9 and P10.

The targets served three purposes at once: to measure the displacements of the beam, to measure the deformation of the neoprene bearings and to measure any settlement of the foundation plinths. By combining these readings, the displacement of the beam alone could be recovered, once the contribution of the bearing deformation and of the plinth settlement had been removed [6]. Two of the targets are of particular interest for this work. P10 is placed at the top of the mid-section, at the level of the slab, and P9 at the bottom chord of the mid-section. The difference between their vertical readings gives the change in length of the section over its full height at the midspan.

For the evaluation of the rotations, the targets are grouped into five cross-sections distributed along the beam: at the support on the plinth A side ($z = 0$ m), at $z = 8.25$ m, at the midspan ($z = 12.8$ m), at $z = 17.35$ m and at the support on the plinth B side ($z = 25.6$ m). The longitudinal coordinate z is measured along the axis of the beam, starting from the support on the plinth A side.



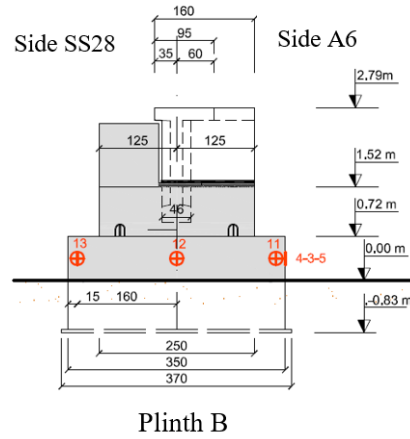
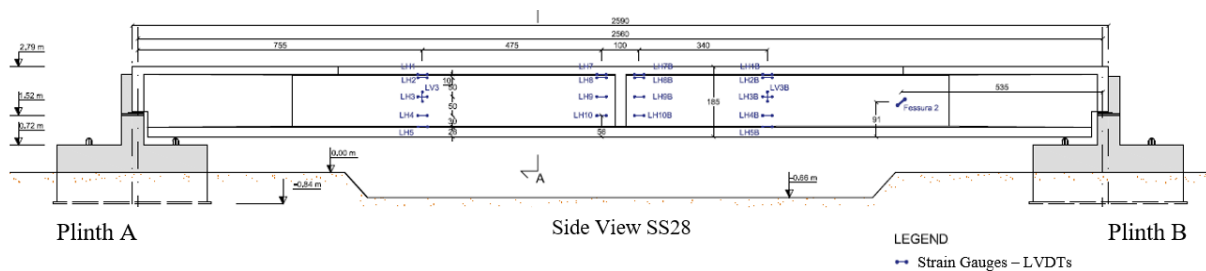


Figure 2.9. Layout of the 24 topographic targets.

2.6.2 LVDT extensometers

The strains in the concrete were followed by means of LVDT extensometers (Linear Variable Differential Transformers). Eighteen of these sensors were mounted in the longitudinal direction and grouped into four vertical alignments, one in each of four cross-sections located at z equal to 7.5, 12.2, 13.4 and 16.8 m. Within every alignment the sensors are spread over the height of the section, so that the profile of the longitudinal strain across the height can be reconstructed at each of the four positions [6]. Two additional extensometers, denoted LV3 and LV3B, were placed in the vertical direction at mid-height of the web and paired with the corresponding longitudinal sensors. The measuring base of the LVDTs is 400 mm and the sampling rate is 10 Hz. Their arrangement is shown in Figure 2.10.



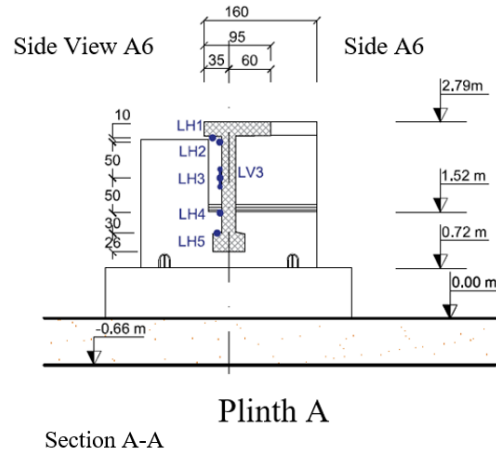


Figure 2.10. Layout of the LVDT extensometers in the monitored sections.

2.6.3 Strain gauges

Two induction strain gauges were bonded to the main tensile reinforcement of the lower chord, on one of the 20 Φ 28 bars [6]. The first gauge, SGSX, is located at roughly one third of the span, at $z = 5.60$ m, and the second, SGMZ, at the midspan, at $z = 12.8$ m. Both gauges recorded the strain in the bar at a sampling rate of 10 Hz, and they were intended to detect the onset of yielding in the reinforcement. Their positions are shown in Figure 2.11.

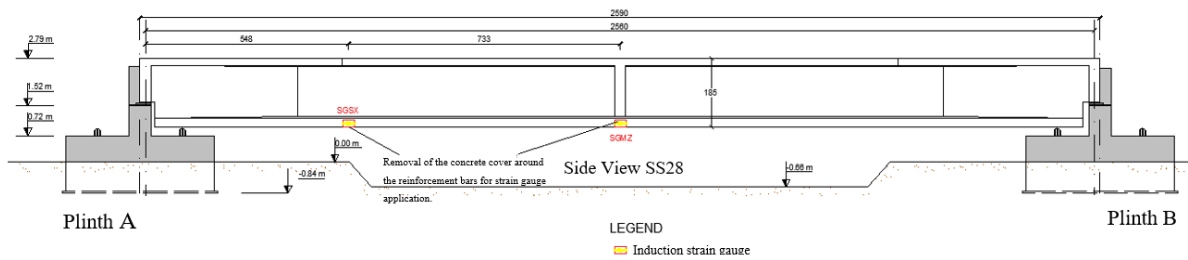


Figure 2.11. Position of the strain gauges SGSX and SGMZ on the bottom chord.

2.7 Load test results

This section presents the results of the load test. The displacements and the rotations are expressed in the reference system of Figure 2.12, in which x is the transverse direction, y the vertical direction and z the longitudinal direction, and a rotation is taken as positive when it is counterclockwise about the axis considered [6]. The collapse mode is described first, followed

by the topographic measurements, the strains recorded by the LVDT extensometers and the strains in the reinforcement.

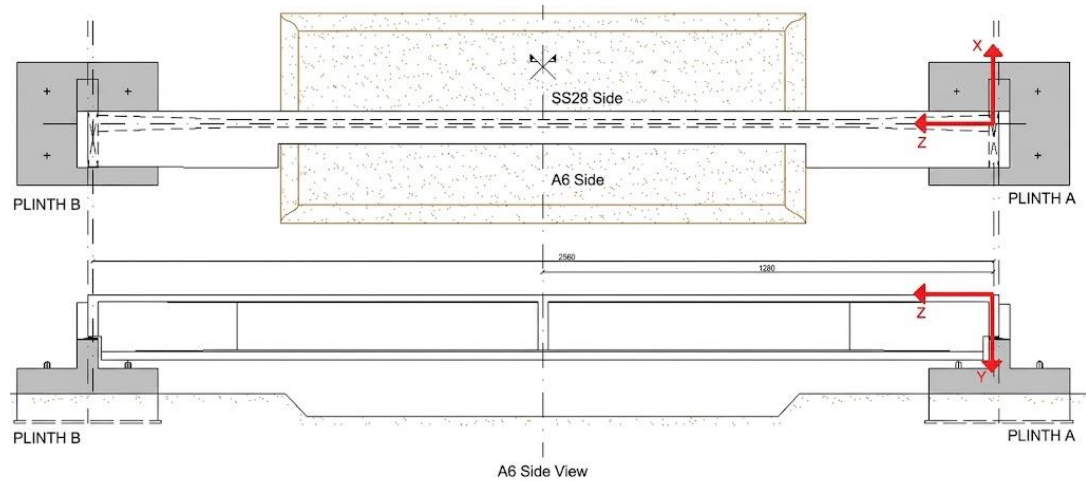


Figure 2.12. Reference system for displacements and rotations.

2.7.1 Collapse mode

The way in which the beam failed was governed by the asymmetry of its cross-section. In spite of the slab cut and the repositioning of the ballast, the response retained a component of skew bending, with a neutral axis that was clearly inclined to the horizontal. The test reached a total load of 1260 kN during the placement of the third layer of ballast, with a vertical displacement at the midspan of about 9 cm [6].

At this load the beam failed by local crushing of the concrete of the slab. The crushed zone is on the side of the slab cut, about 1.50 m from the midspan crossbeam. On the opposite side of the section, the 20 Φ 28 bars of the lower chord had already reached yielding. The final state therefore combined a vertical displacement of about 9 cm at the midspan with a horizontal displacement of comparable magnitude. Because the response coupled skew bending with a limited ductility, the precise instant of collapse was difficult to anticipate [6].

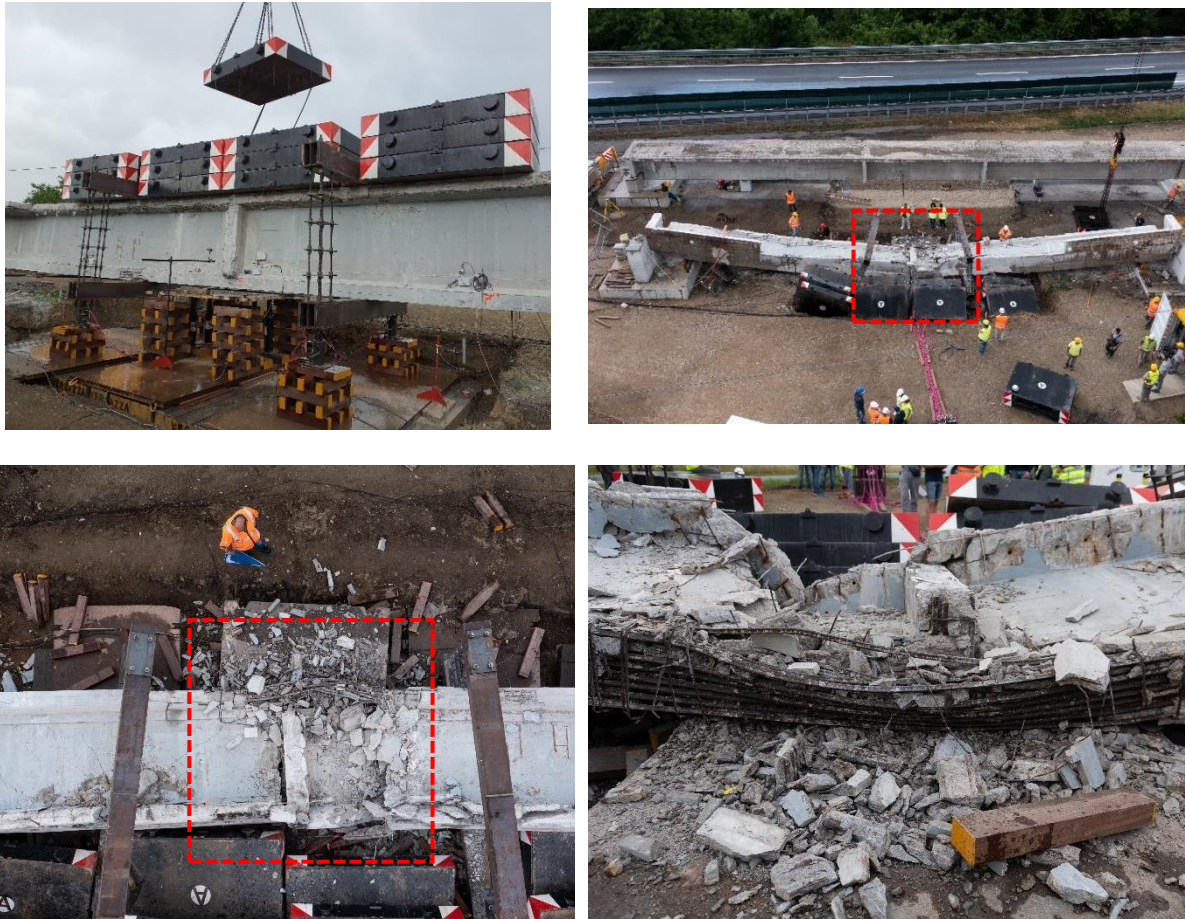


Figure 2.13. Photograph representation of the collapse by crushing of the slab concrete near the midspan.

2.7.2 Topographic measurements

The targets provided the transverse, the vertical and the longitudinal displacement at each loading step. The vertical displacements given here are absolute values, obtained after the displacements of the supports have been removed [6]. They are reported for the pairs of targets P10 and P9, P8 and P8', P7 and P7', P6 and P6', P1 and P1', and P2 and P2', as shown in Figure 2.14.

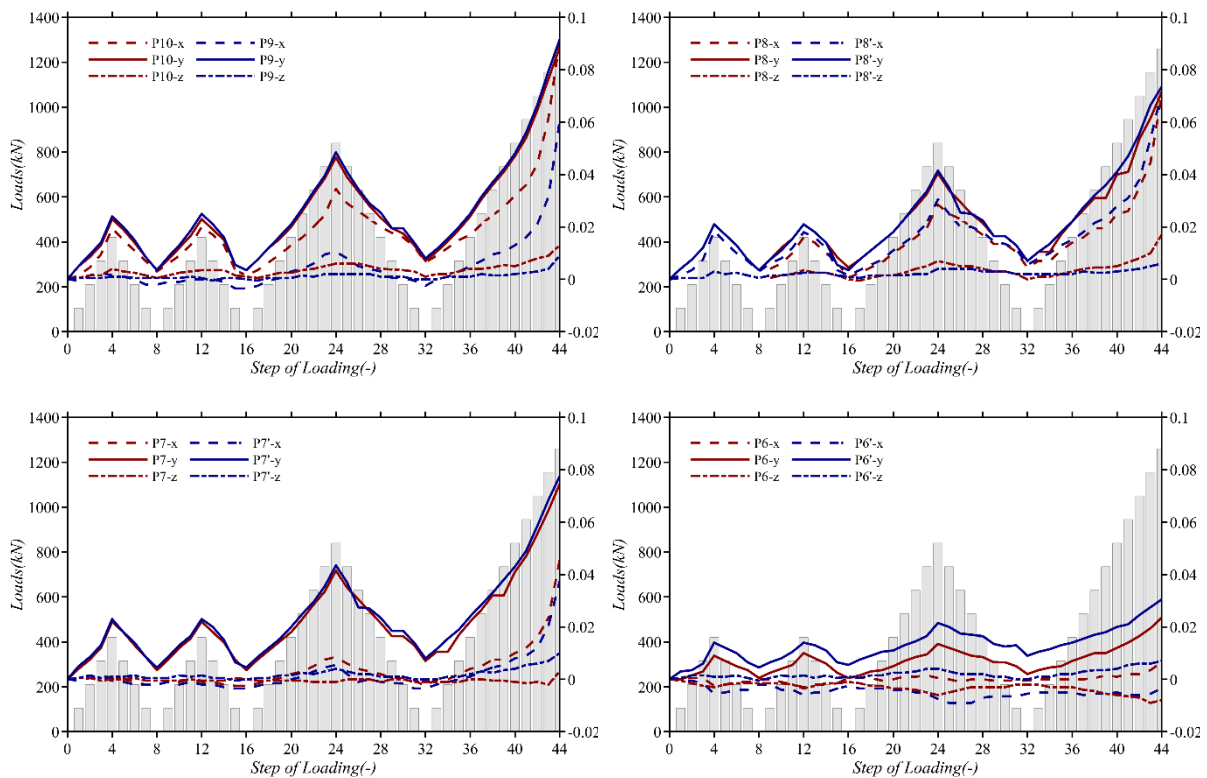
In the transverse direction, the targets at the level of the slab, namely P10, P8 and P8', moved more than the targets placed lower on the beam, P9, P7 and P7'. This difference between the upper and the lower targets reveals a torsion superimposed on the skew bending, an effect that grew more marked in the final stages of the test. During the first two cycles, the transverse displacement of the midspan target P10 was of the same order as its vertical displacement. This

component was reduced once the timber shims had been inserted at the supports, as described in Section 2.5. The maximum vertical displacement at the midspan was about 9 cm.

In the longitudinal direction, the displacements recorded by all the targets stayed close to 1 mm for the whole duration of the test.

The flexural rotations were evaluated in the five cross-sections introduced in Section 2.6, at z equal to 0, 8.25, 12.8, 17.35 and 25.6 m. As the load increased, these rotations took on a pattern that is antisymmetric about the midspan. During the fourth cycle a sharp rise appears between the sections at 12.8 m and at 17.35 m, in the region where the slab later crushed, and there the rotation reaches about half a degree. The flexural rotations are shown in Figure 2.15.

The torsional rotations were evaluated in the same five cross-sections. They keep one and the same sign along the beam throughout the test, which confirms the torsional component of the response. The torsional rotations are shown in Figure 2.16.



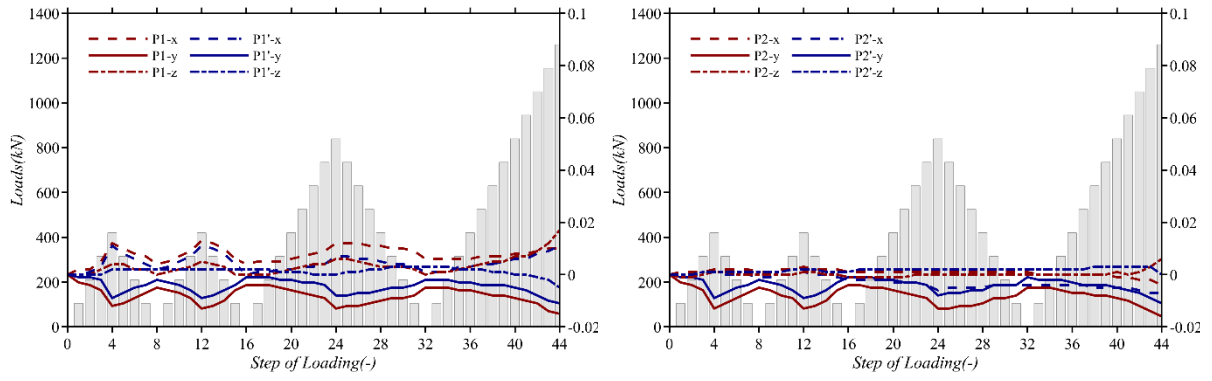
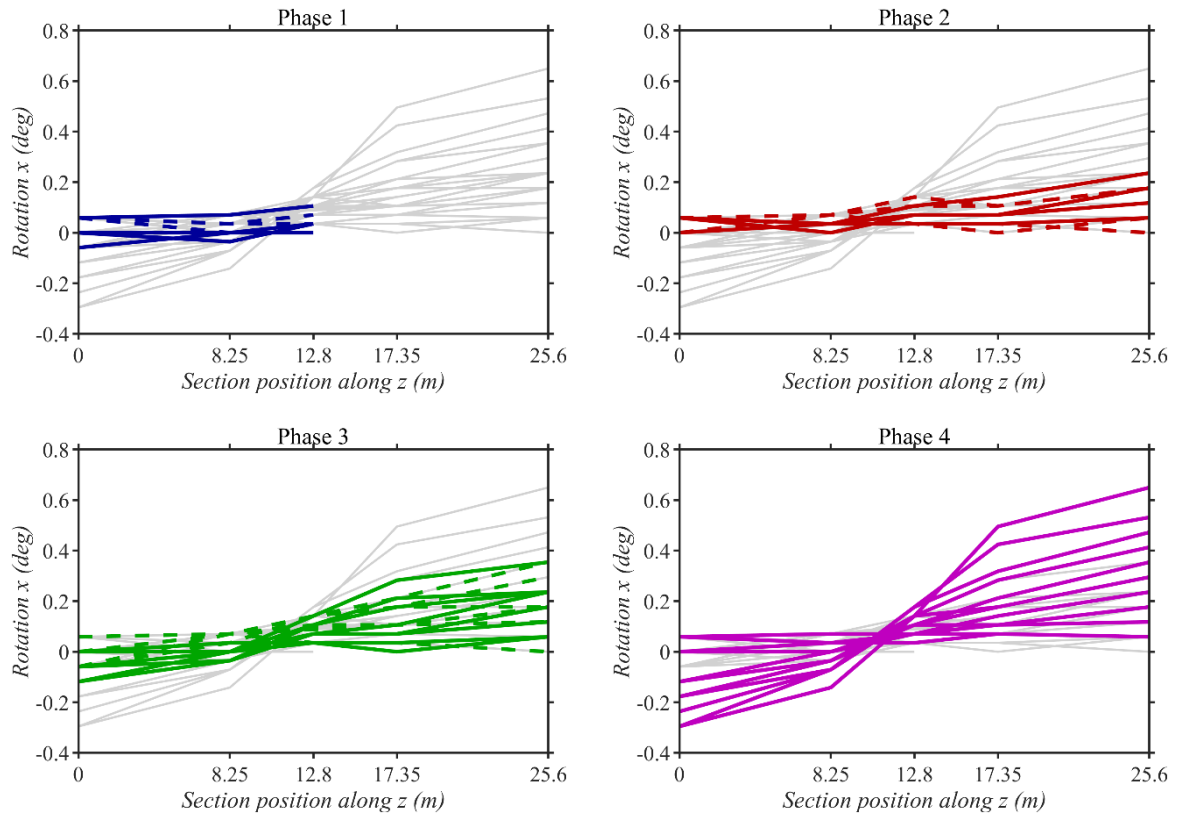


Figure 2.14. Displacements of the target pairs against the loading step.



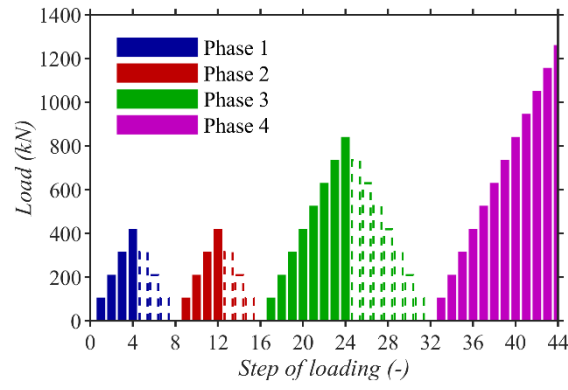
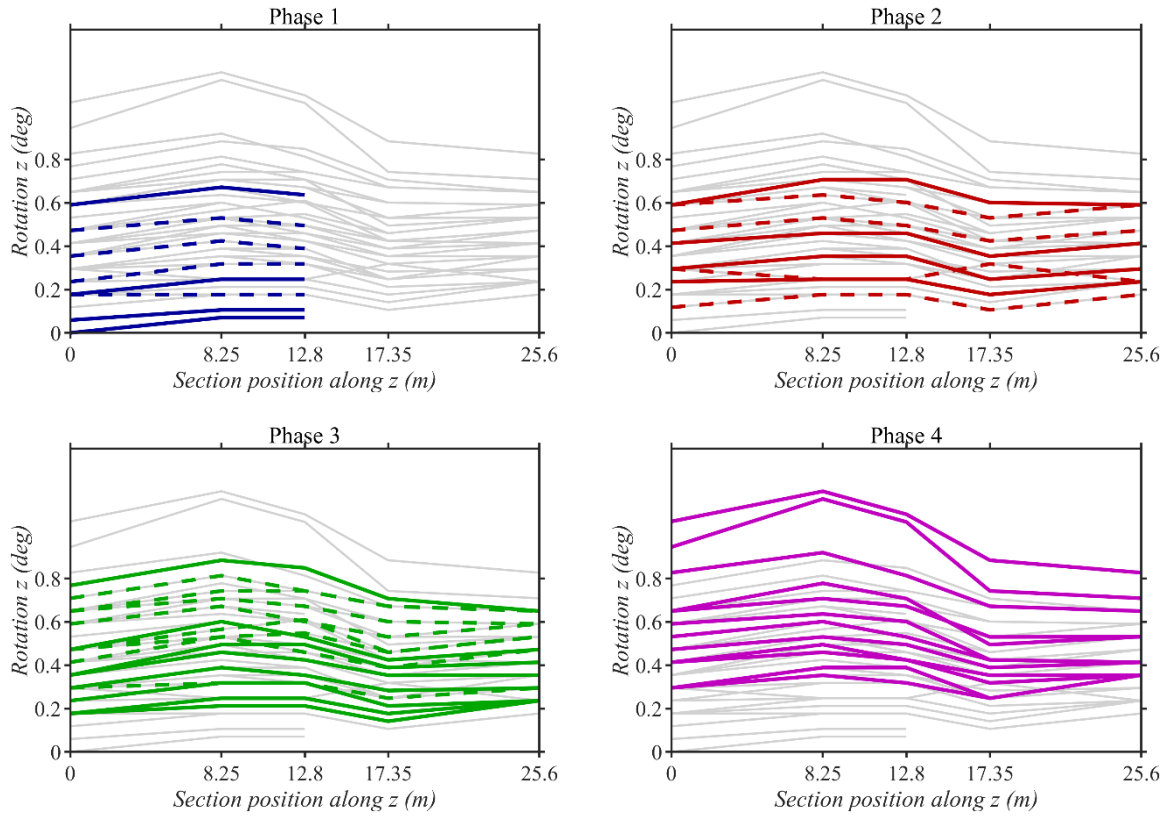


Figure 2.15. Flexural rotations in the five monitored sections for each 4 phases.



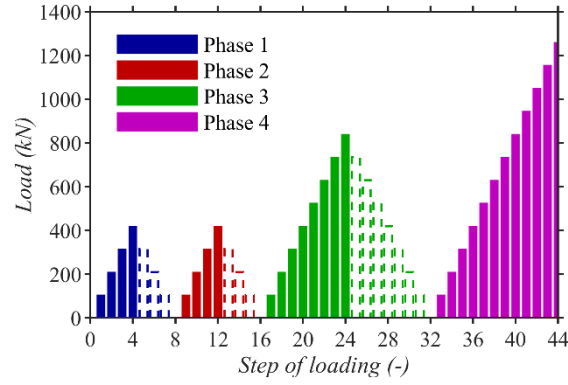
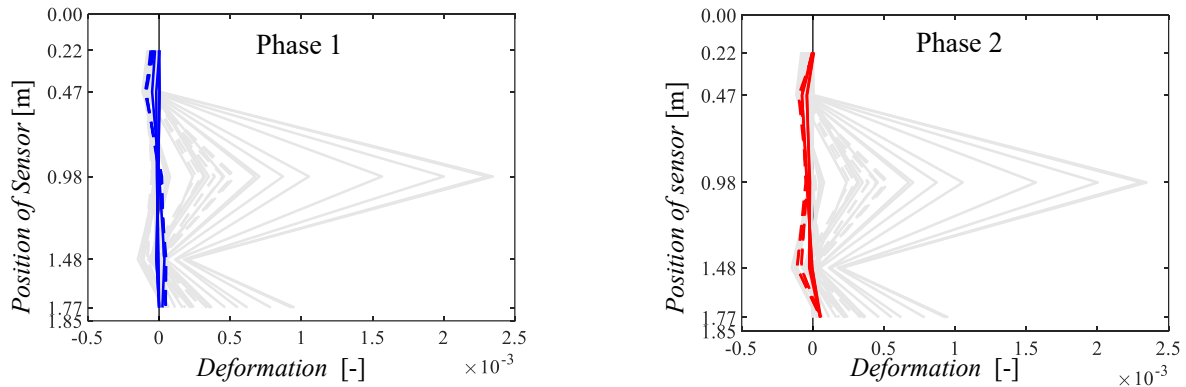


Figure 2.16. Torsional rotations in the five monitored sections for each 4 phases.

2.7.3 LVDT measurements

The LVDT extensometers describe how the longitudinal strain is distributed over the height of the four instrumented sections, with elongation taken as positive and shortening as negative [6]. The clearest results come from the sections at $z = 7.5$ m and $z = 16.8$ m. In both of them the strain reaches a peak near the mid-height of the section, at the transition between the lower chord and the slenderer central web, and the same tendency, though weaker, can be seen in the other two sections. The section at $z = 16.8$ m shows a further feature, a strain profile that crosses zero at two points over the height. This shape is typical of a section in skew bending. When the neutral axis is inclined, and therefore not orthogonal to the plane of the applied moment, it intersects the outline of the section at two separate points, and the longitudinal strain consequently vanishes at two distinct levels. The strain profiles measured in the four sections are shown in Figure 2.17- Figure 2.20.



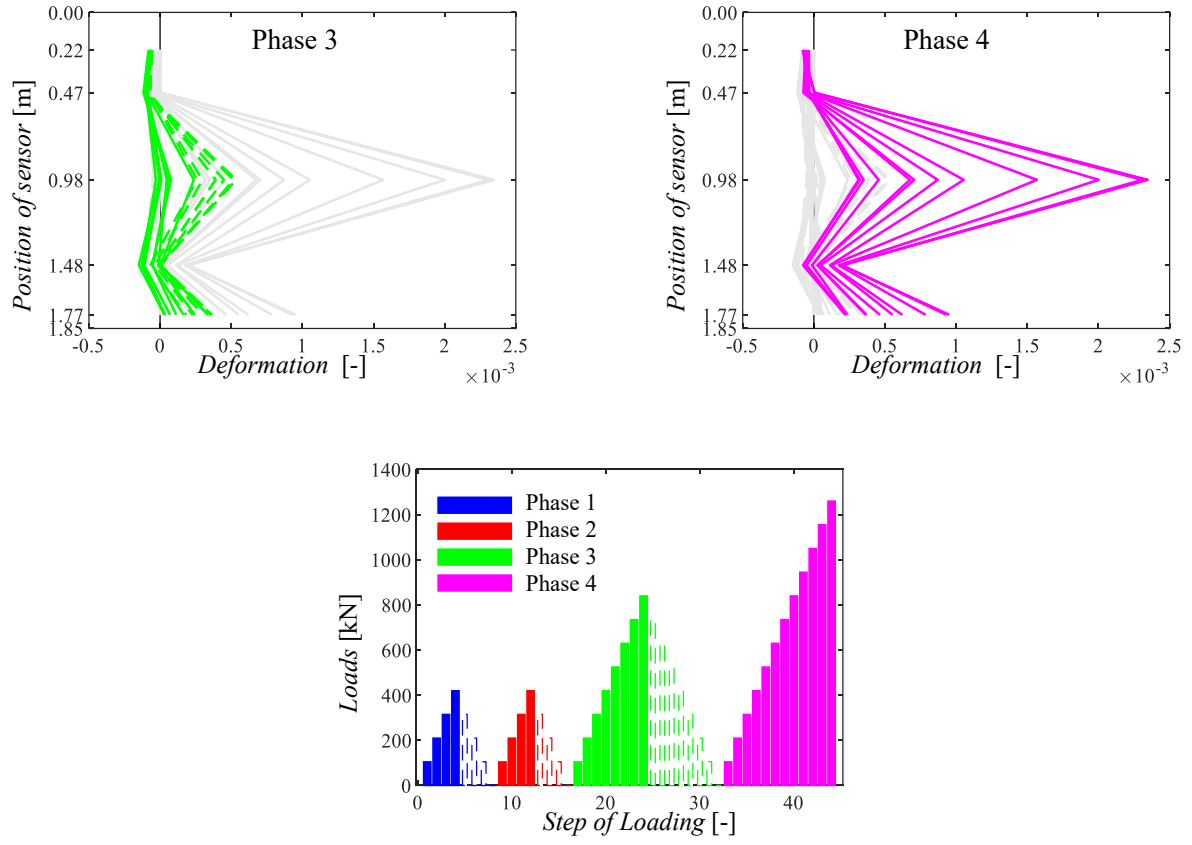
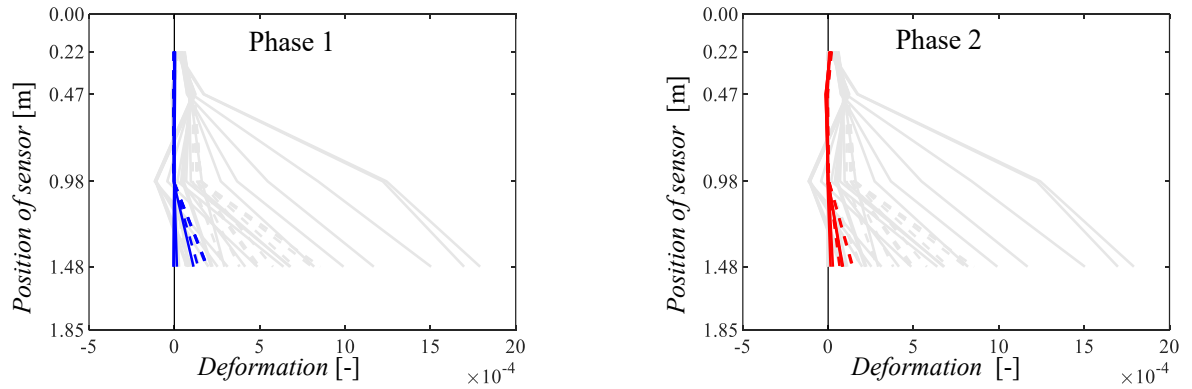


Figure 2.17. Longitudinal strain profiles at section $z=7.5\text{m}$, over the height of the monitored sections for all 4 phases.



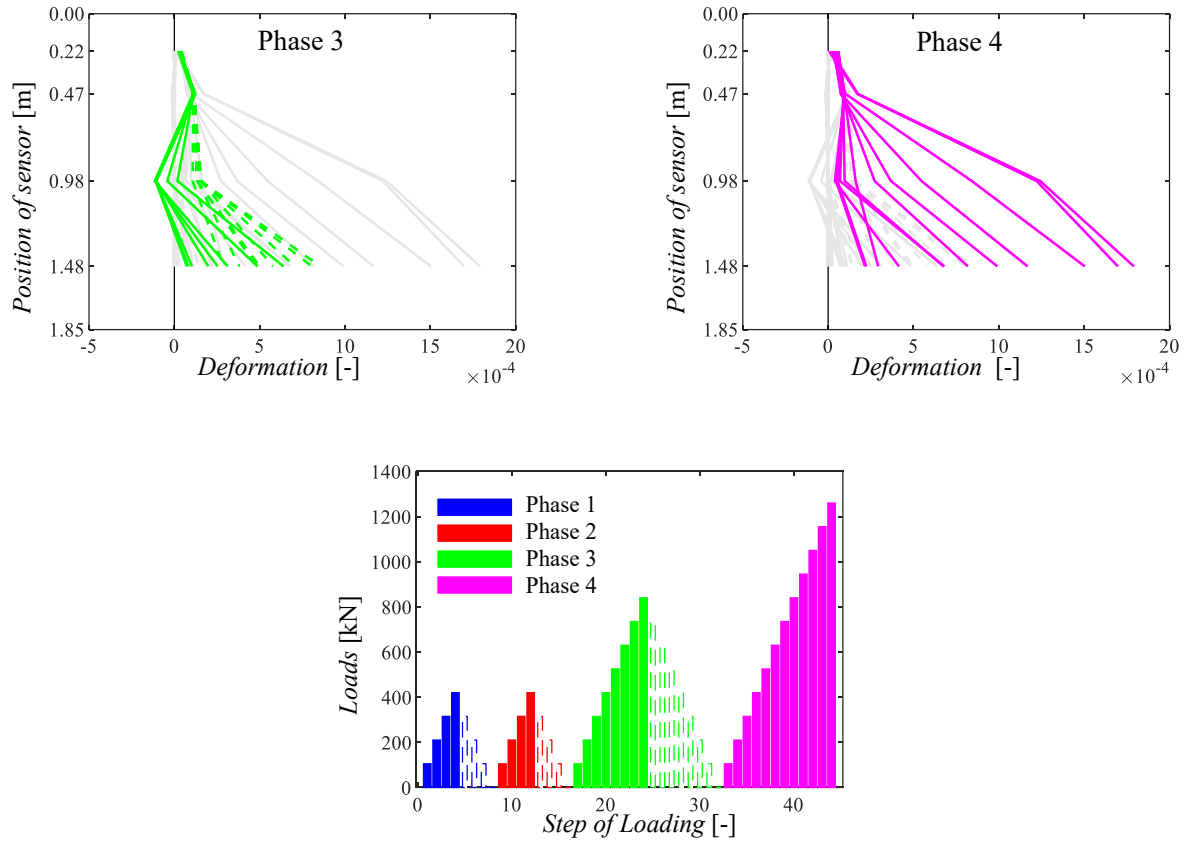
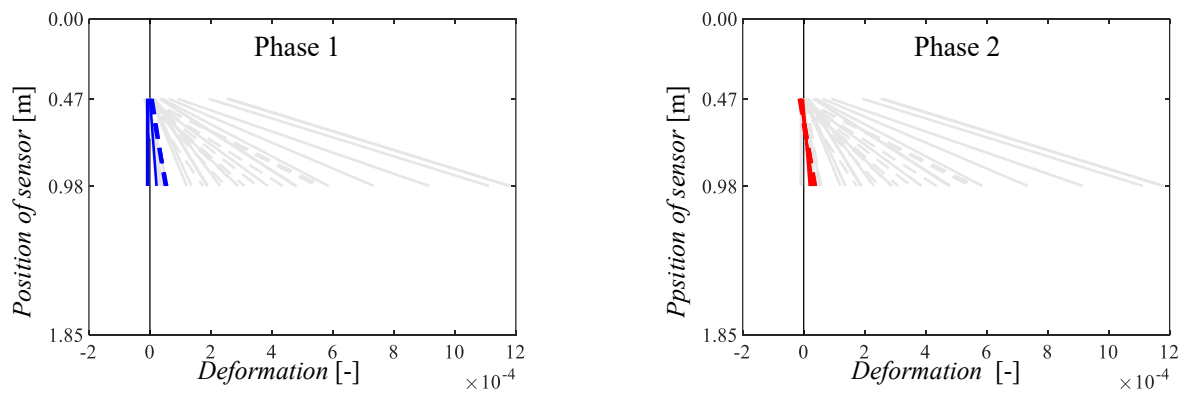


Figure 2.18. Longitudinal strain profiles at section $z=12.2\text{m}$, over the height of the monitored sections for all 4 phases.



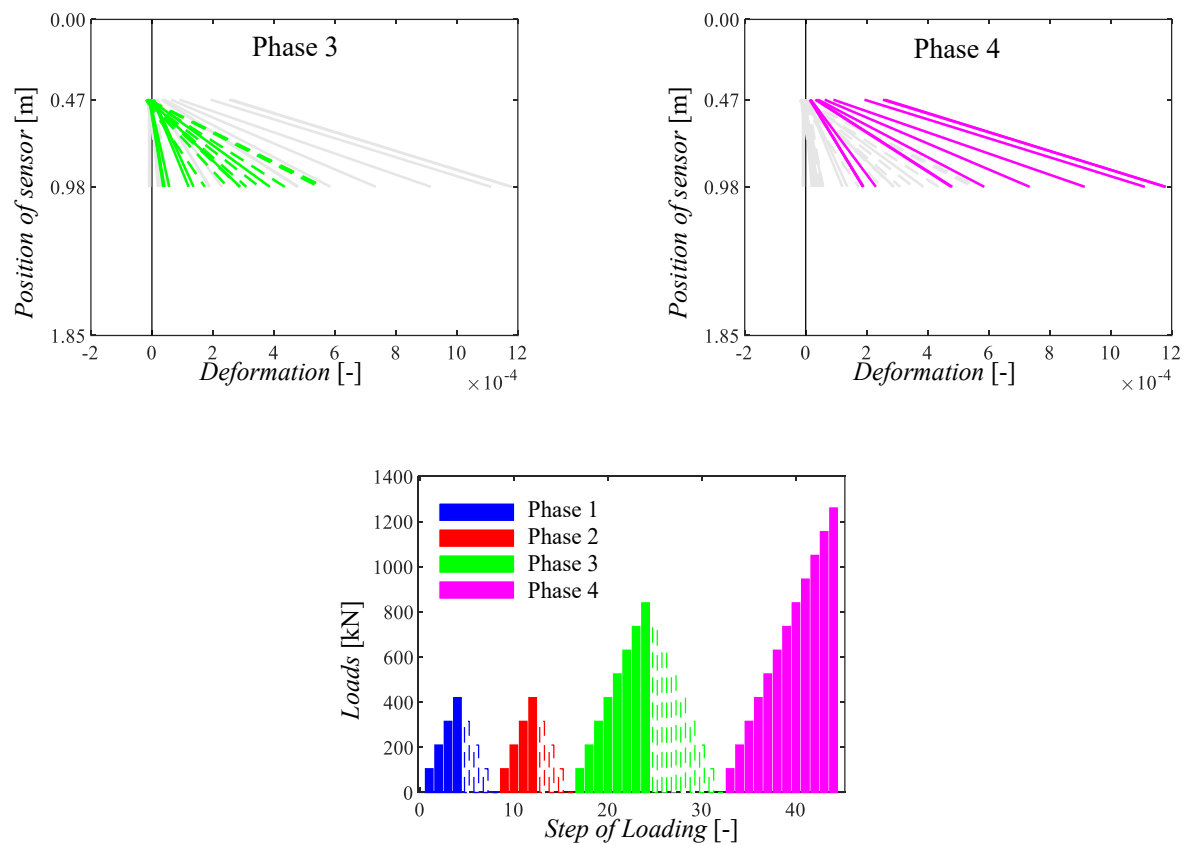
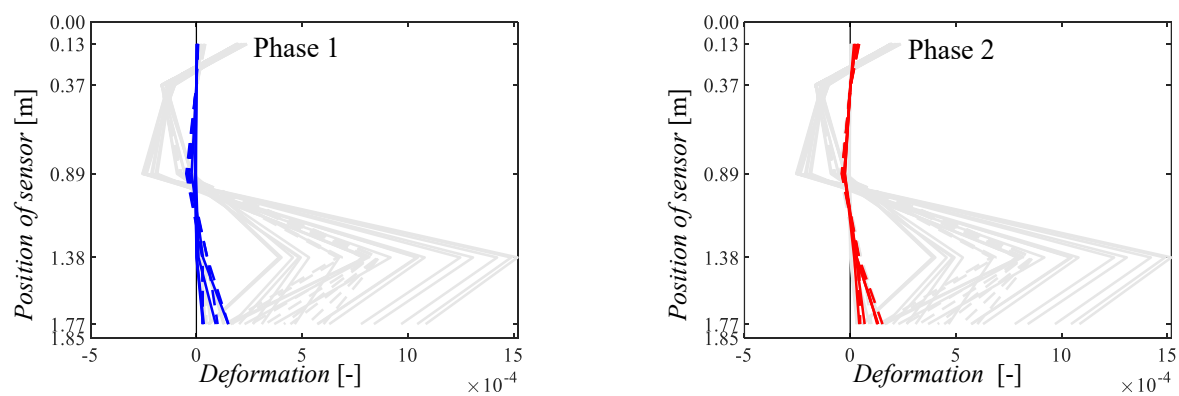


Figure 2.19. Longitudinal strain profiles at section $z=13.4\text{m}$, over the height of the monitored sections for all 4 phases.



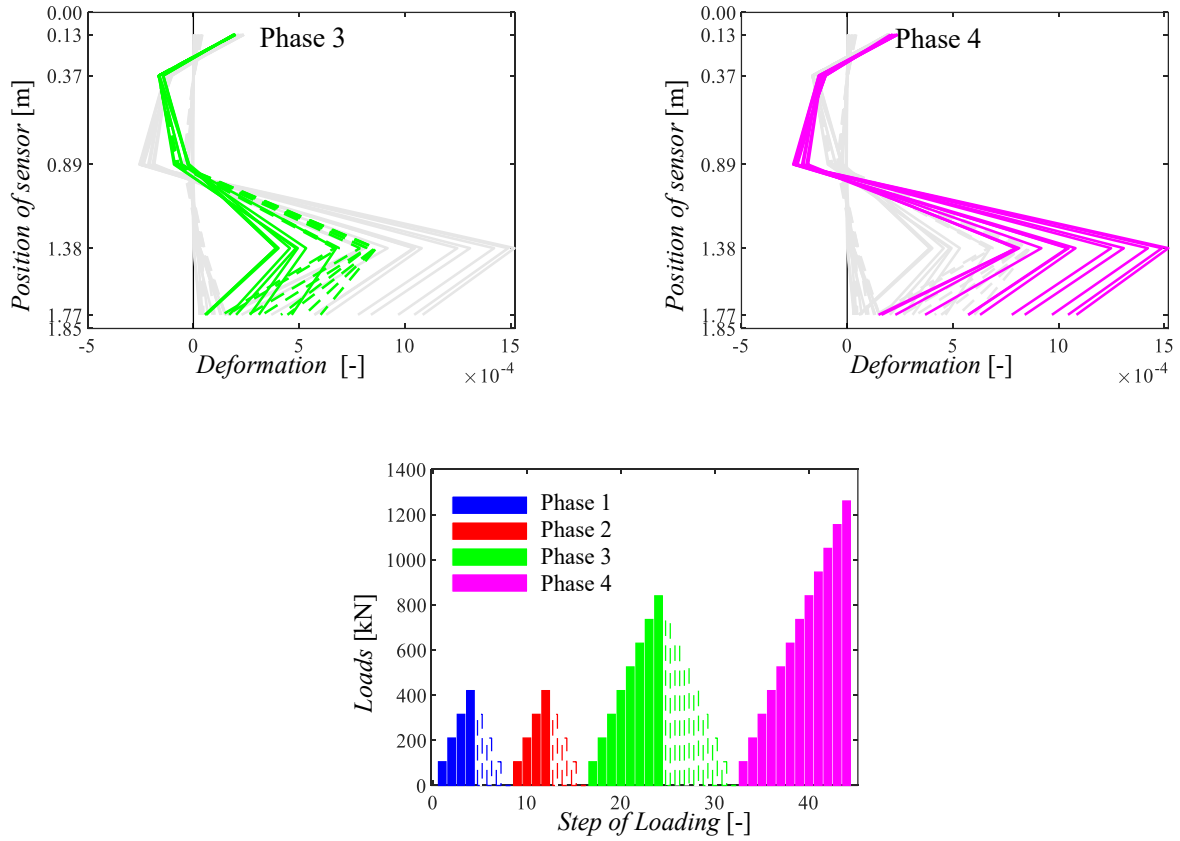


Figure 2.20. Longitudinal strain profiles at section $z=16.8\text{m}$, over the height of the monitored sections for all 4 phases.

2.7.4 Reinforcement strains

The two induction strain gauges recorded the strain in a bar of the lower chord, at $z = 5.60\text{ m}$ for SGSX and at the midspan, $z = 12.8\text{ m}$, for SGMZ. At the midspan gauge the steel reached its yield strain and then went beyond it at the start of the fourth loading cycle [6]. This finding agrees with the yielding of the lower bars observed after the collapse and reported in Section 2.7.1. The strains measured by the two gauges are shown in Figure 2.21.

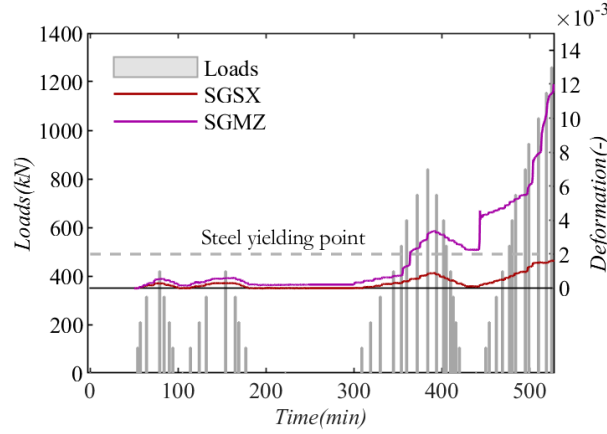


Figure 2.21. Strains measured by SGSX and SGMZ during the test.

2.8 Laboratory tests

After the test, samples of concrete and of reinforcing steel were taken from several regions of the beam and tested to failure in the laboratory of the DISEG [6]. The representative values of the properties were then obtained by a statistical analysis of the test results. This section sets out the statistical method, the results measured on the concrete cores and the results measured on the reinforcement. The mean values reported here are those assigned to the materials of the numerical model in Chapter 3, in which the complete reinforcement of the original drawings is reproduced.

2.8.1 Statistical method

Each material property X is treated as a random variable with a lognormal distribution [25], [30]. The parameters of the distribution are estimated by the maximum likelihood method applied to the variable Y , defined as the natural logarithm of X . Once the mean and the standard deviation of Y , denoted μ_Y and σ_Y , are known, the mean μ_X , the standard deviation σ_X and the coefficient of variation V_X of the property follow by inverting the relations:

$$\mu_Y = \ln \mu_X - 0.5 \ln(1 + V_X^2) \quad (2.1)$$

$$\sigma_Y = [\ln(1 + V_X^2)]^{0.5} \quad (2.2)$$

The characteristic value of the property is then obtained as:

$$X_k = \mu_X \exp(k_f V_X) \quad (2.3)$$

where the coefficient k_f fixes the fractile of the distribution. For the strengths the characteristic value is taken at the 5 % fractile, with k_f equal to -1.645 , while for the ultimate strain of the steel it is taken at the 10 % fractile, with k_f equal to -2.326 . The design values are derived from the characteristic values through the partial factors of EN 1992-1-1 [25]. This procedure accounts both for the natural scatter of each property and for the uncertainty introduced by the limited number of samples [25], [30].

2.8.2 Concrete cores

A total of 14 cores were extracted from the beam. Seven of them come from the web of the precast beam (C1, C2, C3+, C3+2, C7, C9 and B) and seven from the cast-in-place slab (C4, C5, C6, C8, A, R2 and B2). The cores A and B2 were taken from the region of the slab that crushed during the test [6].

The compression test on each specimen gives the maximum stress that it reaches, $\sigma_{\max}$. The in-situ cylindrical compressive strength $f_{c,is}$ is obtained from this stress according to EN 13791 [39]:

$$f_{c,is} = F_d k_f \sigma_{\max} \quad (2.4)$$

Here F_d is a factor that accounts for the disturbance caused by coring, taken equal to 1.00 when $\sigma_{\max}$ exceeds 40 MPa, and k_f is a factor that accounts for the slenderness of the specimen:

$$k_f = 0.83 + (l/d - 1)(1 - 0.83) \quad (2.5)$$

in which l and d are the length and the diameter of the specimen. The stabilized secant elastic modulus $E_{c,s}$ could be measured only on the cores A and B, whose slenderness was suitable for the test.

Table 2.3 collects the statistical parameters of the in-situ compressive strength, separately for the web and for the slab, together with the elastic moduli measured on cores A and B. The mean compressive strength is 46.05 MPa for the web of the precast beam and 52.13 MPa for the slab, while the measured elastic modulus is 29626 MPa for the web, from core B, and 37481 MPa for the slab, from core A. These mean values are those adopted for the concrete of the numerical model.

Table 2.3. Statistical parameters of the in-situ cylindrical compressive strength of the concrete, with the elastic moduli measured on cores A and B.

Position	$\mu_{fc,is}$ [MPa]	$\sigma_{fc,is}$ [MPa]	$V_{fc,is}$ [-]	$f_{ck,is}$ [MPa]	$E_{c,s}$ [MPa]
Web (precast)	46.05	2.90	0.063	41.50	29626 (Core B)
Slab (cast-in-place)	52.13	7.15	0.137	41.61	37481 (Core A)

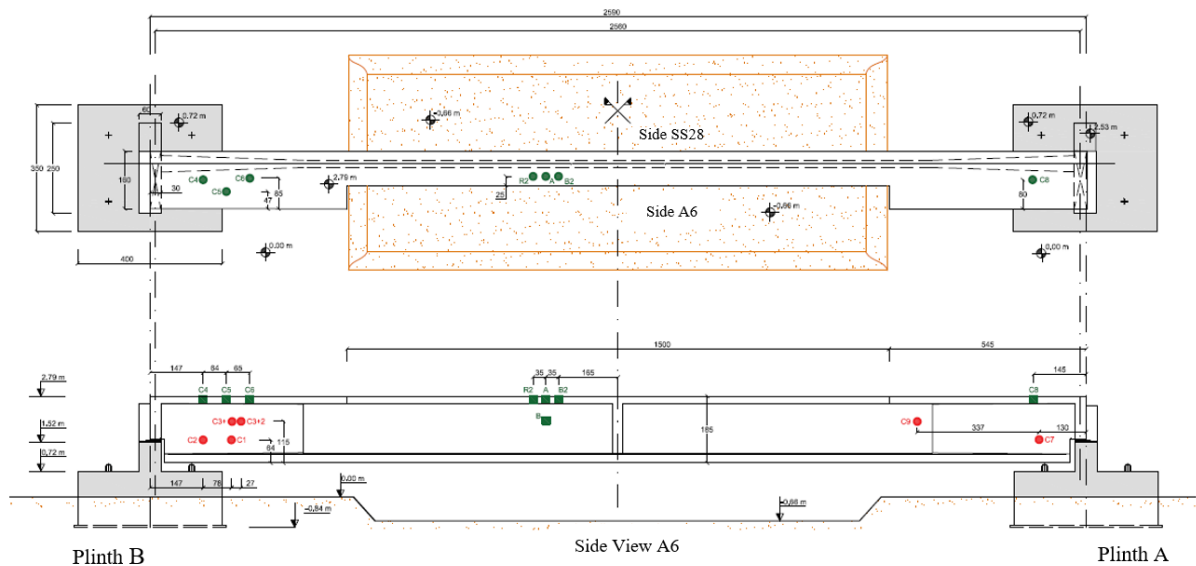


Figure 2.22. Location of the 14 concrete cores.

2.8.3 Reinforcing steel

Ten samples of reinforcing bar were taken from three regions of the beam, denoted B1, B2 and B3 [6]. The samples are ribbed bars with a square cross-section and nominal diameters of 28, 12 and 8 mm. They belong to the longitudinal reinforcement of the lower chord, to the stirrups of the web and to the transverse and longitudinal reinforcement of the slab. The equivalent cross-sectional area $A_{s,eq}$ of each sample was computed from its weight, on the assumption of a steel density of 7850 kg/m^3 .

Table 2.4 reports the statistical parameters of the ordinary reinforcing steel obtained from the tensile tests. The mean yield strength is 397.2 MPa, the mean tensile strength is 567.3 MPa and the mean ultimate strain is 5.81 %. With an elastic modulus of 200 GPa, the corresponding mean yield strain is about 0.20 %.

Given the results of the tensile tests on the four $\Phi 28$ samples of the lower chord, which form the main flexural reinforcement of the beam their yield strength ranges from 387 to 445 MPa and their tensile strength from 577 to 657 MPa, with ultimate strains between 3.7 and 11.8 %. This scatter reflects the different degrees of corrosion of the bars in the three regions. In the numerical model the mean values of Table 2.4 are assigned to the reinforcement, while the complete bar layout of the original drawings is reproduced.

Table 2.4. Statistical parameters of the ordinary reinforcing steel obtained from the tensile tests.

Property	Mean	Standard deviation	Coeff. of variation	Characteristic
f_y [MPa]	397.20	38.96	0.098	338.00
f_t [MPa]	567.25	82.18	0.145	446.60
ϵ_u [%]	5.81	3.33	0.574	1.57

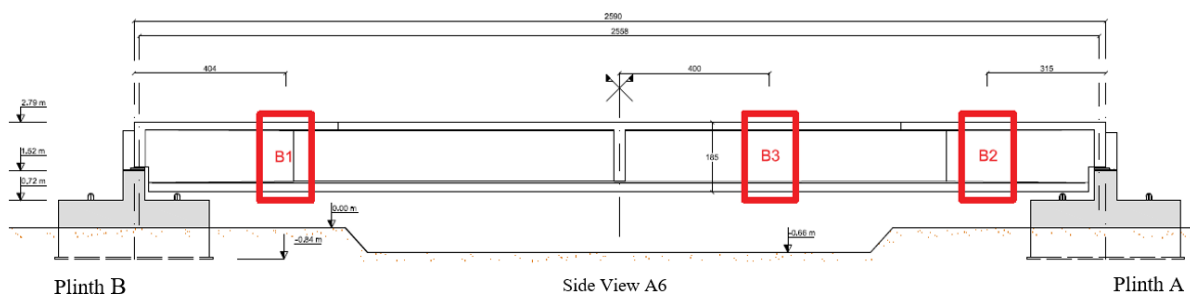


Figure 2.23. Location of the sampling regions B1, B2 and B3.

2.9 Summary

This chapter has presented the load-to-collapse test on the 25.90 m beam, together with the data that the test produced. The element is an edge beam with a strongly asymmetric cross-section, and this feature shaped both the preparation of the test and its outcome. The load was applied by placing cast iron ballast directly on the slab, in successive loading and unloading cycles, up to a total of 1260 kN reached during the third layer. The beam failed by crushing of the slab concrete near the midspan, while the bars of the lower chord had reached yielding, in a mechanism governed by skew bending.

The destructive tests carried out after the experiment provided the mean properties of the concrete and of the reinforcing steel. These properties, together with the geometry, the reinforcement layout and the response measured in terms of displacements, rotations and strains, make up the reference set of data for the next chapter, in which the nonlinear finite element model of the beam is built and its results are compared with the measurements.

Chapter 3. Nonlinear finite element model of the beam

3.1 Introduction to nonlinear analysis

The assessment developed in this thesis belongs to Level 4 of the 2020 Guidelines for existing bridges [3] in which the safety of a structural element is evaluated with refined numerical models validated against experimental evidence. In this chapter I describe the three-dimensional nonlinear finite element model that I developed in ATENA Engineering to reproduce the load test presented in Chapter 2.

Classical methods of structural analysis estimate the internal forces (bending moment, shear and axial force), the displacements, rotations and curvatures, and the support reactions of a structure under simplifying assumptions about the behavior of the materials and of the structure itself. The most common ones are linear elastic analysis, elastic analysis with limited redistribution, rigid-plastic analysis and the strut-and-tie method. They are efficient and well established, both for the design of new structures and for the verification of existing ones. However, they cannot follow the gradual loss of stiffness of a reinforced concrete member once the concrete cracks and the reinforcement yields, and they cannot describe the formation of a collapse mechanism. For a bridge designed to the codes of the 1960s, a linear elastic verification under the traffic loads of the current standards is often impossible to satisfy, even when the real member still has an adequate margin of resistance. A nonlinear analysis removes this limit because it works directly with the constitutive laws of the materials, so the full capacity of the cross-section and the redistribution of internal forces can be exploited [40].

Two sources of nonlinearity are usually distinguished in the theory manual of the program [41]: the nonlinearity of the materials, expressed by stress-strain laws (σ - ϵ) in which the elastic modulus is not constant and plastic strains develop, and the geometric nonlinearity, in which the equilibrium equations are written in the deformed configuration. In the present model only the material nonlinearity is activated.

Nonlinear finite element analyses (NLFEA) of reinforced concrete members are used today for the safety assessment of existing structures, seismic push-over analyses, fire resistance studies, the study of discontinuity regions where the hypothesis of plane sections does not hold, and for realistic estimates of cracking and deflections [40], [42]. The application of interest here is the

first one. A model validated against a collapse test becomes a virtual prototype of the beam, and the same modelling choices can then support Level 4 assessments of similar decks of the national road network. The theoretical background of the constitutive models, of the safety formats and of the validation of NLFEA has been introduced in 0. In this chapter the theory is recalled only where it is needed to justify a specific choice made in the model.

3.2 Steps of the NLFEA

A nonlinear finite element analysis is the simulation of a physical problem described by a nonlinear system of partial differential equations with boundary conditions. The system is solved with the finite element method combined with an iterative solution procedure [41]. Three consecutive phases can be identified in the process.

- Pre-processing: The structure is first idealized into a mechanical model, which may be one-, two- or three-dimensional, and the mechanical model is then discretized into finite elements. This phase includes the definition of the geometry, of the boundary conditions, of the constitutive laws of the materials, of the mesh, of the load history and of the iterative method with its convergence criteria. The quality of the final result depends mainly on the decisions taken here.
- Processing: The equations of all the elements are assembled into a global system that accounts for the interaction between elements through their shared nodes. The system is solved with the chosen iterative method, providing the nodal displacements at each step.
- Post-processing: The results are visualized and interpreted in terms of displacements, reactions, stress and strain fields and the cracking state.

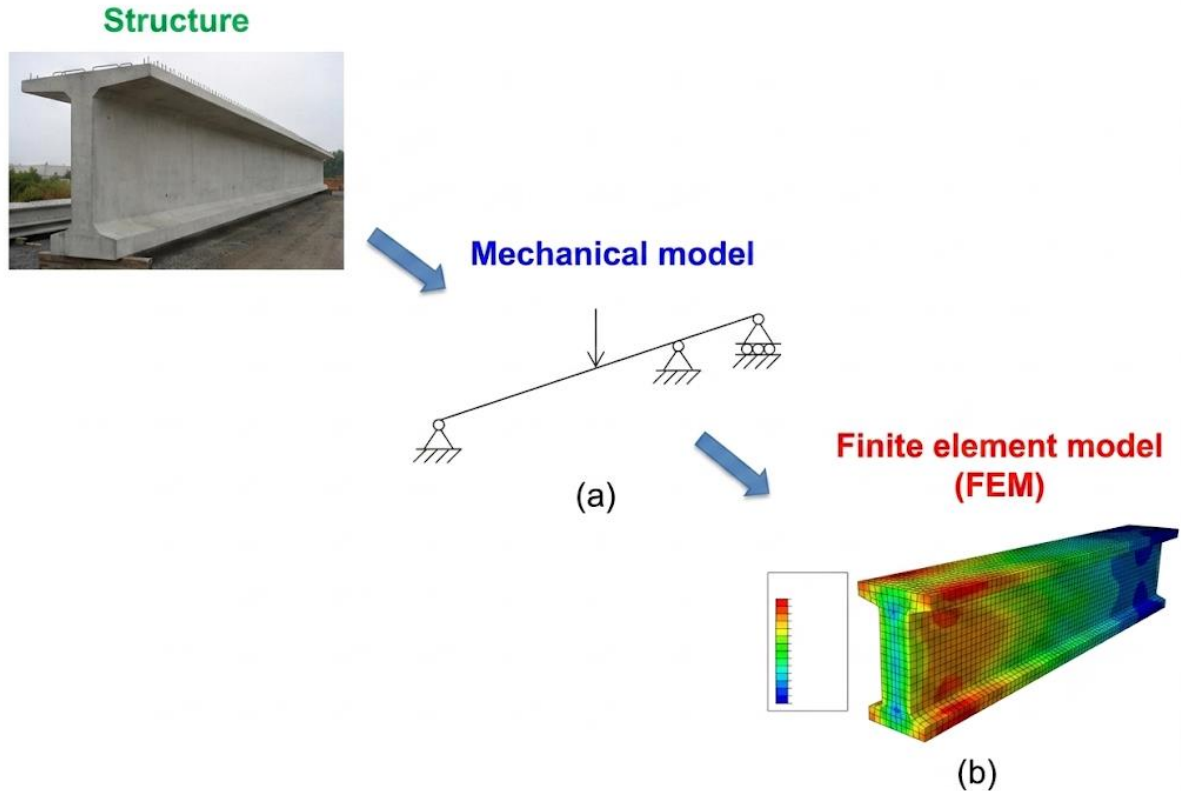


Figure 3.1. Idealization of the structure into a mechanical model and discretization into a finite element model.

The results of a nonlinear analysis are physical quantities rather than abstract numbers. The complete validation against the experimental measurements is presented in Chapter 5. A further point deserves attention when an analysis stops. A physical failure of the model occurs when convergence is lost because the ultimate strain has been reached in some region and a global collapse mechanism has formed. A purely numerical failure occurs when convergence is lost at strain levels far from the ultimate ones, or at strains larger by orders of magnitude [41]. Distinguishing the two cases is essential before interpreting the last converged step as the resistance of the beam, and this distinction is applied in Chapter 4.

3.3 Solution strategy

The solution strategy is the complete set of hypotheses adopted to solve the mechanical problem with a nonlinear finite element model. These hypotheses concern the equilibrium between external and internal forces, the kinematic compatibility of displacements and strains, and the constitutive laws of the materials. Choices such as the type and size of the finite elements, the iterative method and the convergence criteria belong to the solution strategy and

have a direct influence on the result of the analysis [41]. For this reason, they are documented one by one in the following sections.

The mechanical problem is expressed by the system of equilibrium equations of the discretized structure:

$$\lambda p - f(q) = 0 \quad (3.1)$$

which holds in the linear and in the nonlinear case and can be rewritten as:

$$K(q) q - f(q) = 0 \quad (3.2)$$

where λ is the load multiplier, p is the vector of the external nodal loads, q is the vector of the nodal displacements, $f(q)$ is the vector of the internal nodal forces as a function of the displacements, and $K(q)$ is the stiffness matrix of the system, which in the nonlinear case also depends on the displacements. At each load level the system produces a residual, given by the imbalance between external and internal forces, which the iterative method progressively cancels. The method adopted in this work is the Newton-Raphson procedure, described together with the convergence criteria in Section 3.13.

3.4 The ATENA Engineering software package

The ATENA package, developed by Červenka Consulting, is developed for nonlinear finite element analyses of reinforced concrete structures [43]. The Engineering line of the package, which I used in this thesis, consists of two programs. ATENA Pre is the pre-processor, in which the geometry, the materials, the mesh, the boundary conditions and the load history are defined [44]. ATENA Studio runs the nonlinear solution and provides the post-processing tools [45]. The versions used for the model of the 25.90 m beam are ATENA Pre 2025 and ATENA Studio 2025, with static analysis in three-dimensional space.

The pre-processor presents the model in a central graphic window, where the user can adjust the viewpoint, rotation, and zoom level using mouse controls similar to standard CAD environments. Visualization settings can be customized by modifying parameters such as object transparency, line thickness, color, zoom behavior, and the visibility of specific entities

and their associated vectors. Above the model window, the main menu and the modelling/viewing toolbars provide access to all pre-processor functions, including geometry and material definition, analysis settings, and frequently used commands for creating, editing, and inspecting model entities. On the left, the Input Data Tree guides the user through the pre-processing workflow, from geometric modelling through mesh configuration to load history specification, while on the right the Layers window allows assigning entities to different layers so that parts of the model can be grouped, temporarily hidden, or protected from accidental modification. Below the workspace, a multi-tab window includes a Data Table, which summarizes the properties of the currently active category (for example, coordinates of geometric objects or key material parameters) and displays messages and error logs during mesh generation, as well as a Script History, where all user actions are recorded as script commands that can be copied, edited, and re-executed to automate model creation. Finally, at the very bottom of the interface, a log window reports each program operation in chronological order, providing a detailed record of the pre-processing session [44].

In ATENA, the finite element model is constructed through a strictly sequential workflow to ensure that no definition stage is inadvertently omitted. The modelling steps are therefore executed in the following order: definition of the geometry (points, then lines, surfaces and volumes); definition of the materials; creation and assignment of the structural element types; creation and assignment of the reinforcement; creation of springs and interfaces; definition of the finite element mesh; definition of the analysis, that is load cases, boundary conditions, monitors, solution parameters and tasks. The structure of this chapter follows the same sequence.

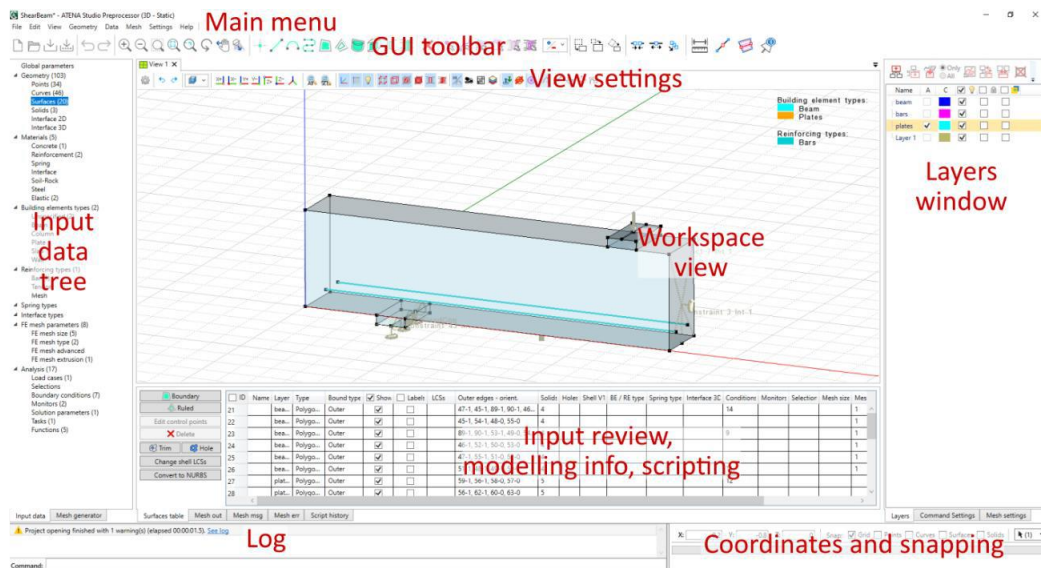


Figure 3.2. Graphical interface of ATENA Pre, adapted from the user's [44]

ATENA Studio performs the solution and shows, in real time, the diagrams of the monitored quantities, the convergence indicators of each iteration and the deformed shape of the structure. This allows the progress of the analysis to be checked and the run to be stopped if it develops incorrectly. After the run, the same interface gives access to the post-processing: the monitored quantities can be plotted from the first to the last step, and the deformed shape, the stress and strain fields and the cracking state can be displayed for each step.

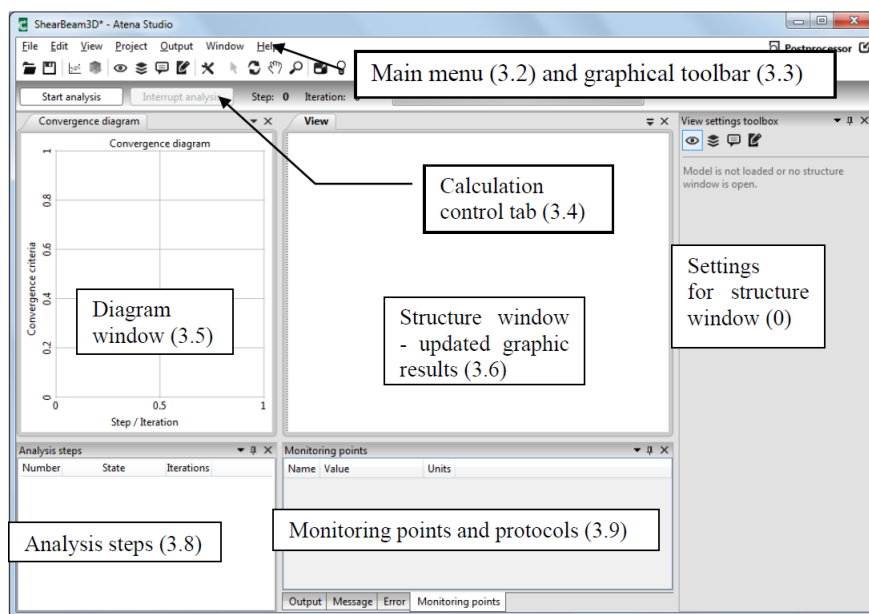


Figure 3.3. Graphical interface of ATENA Studio, adapted from the user's manual [45]

3.5 Model geometry

The geometric model reproduces the beam tested at the test field, whose geometry was reconstructed in Chapter 2 from the original design documents and from the survey carried out on site [6]. A first decision concerned the drawing environment. The cross-section of the edge beam is asymmetric and changes shape along the span, so drawing it point by point inside ATENA Pre would have been slow and prone to errors. I therefore built the complete geometry in AutoCAD and imported it into ATENA as a DXF file. AutoCAD offers precise coordinate control and a layer structure that ATENA preserves during the import, so every group of entities arrives in the pre-processor already separated by layer structural function.

A general rule guided the preparation of the geometry: the geometry must serve the mechanical model. The points and lines where constraints and loads will be applied must belong to the structure, so that they become part of the finite element mesh and can transmit the constraint or load function. In addition, I prepared the solid regions so that each final volume is bounded, where possible, by six faces. This choice simplifies the definition of the volumes in ATENA and favors the generation of a structured hexahedral mesh, which is the element type adopted in this model (Section 3.9). Care was also taken to avoid duplicate or overlapping surfaces, which cause meshing errors and numerical instabilities.

3.5.1 Geometry of the beam

The work in AutoCAD followed an ordered sequence: definition of the drawing layers; drafting of the typical cross-sections; positioning of the sections at their correct longitudinal coordinates; creation of the longitudinal reference lines connecting the sections; adjustment of the reference coordinate system; export in DXF format. The typical cross-sections capture the points of the span where the shape of the precast beam changes, in particular the end regions with the Gerber saddles and the central region of constant section.

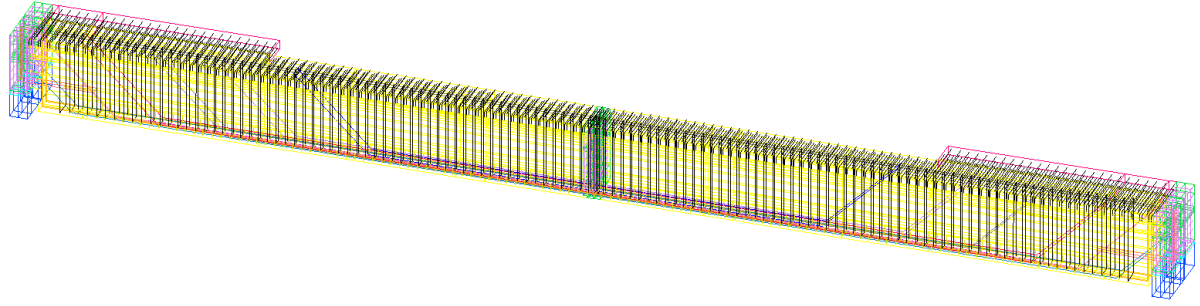


Figure 3.4. Geometry of the beam prepared in AutoCAD before the export to DXF.

The DXF file was imported through the menu path File, Import, DXF. After importing, the model consists only of joints and lines, organized in the original layers. Inside ATENA Pre, I checked the connectivity of the imported entities, closed the few residual gaps, and then generated the surfaces with the command Surface from boundary curves, selecting for each surface the closed set of lines that bounds it. The volumes were generated in the same way from closed sets of surfaces, with the command Create Solids. At the end of this phase the three-dimensional geometry of the precast beam was complete.

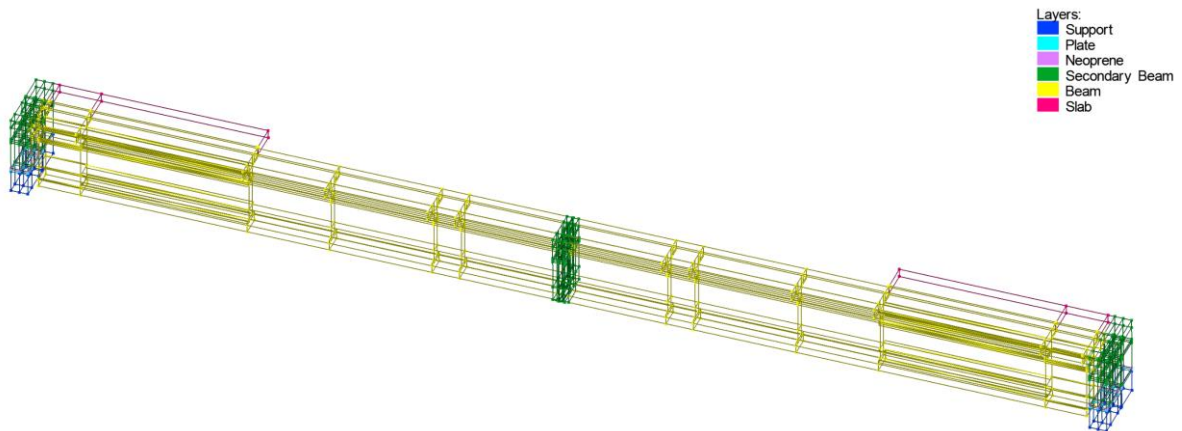


Figure 3.5. Imported model of joints and lines into ATENA Pre, which representing the beam and the auxiliary elements of the test.

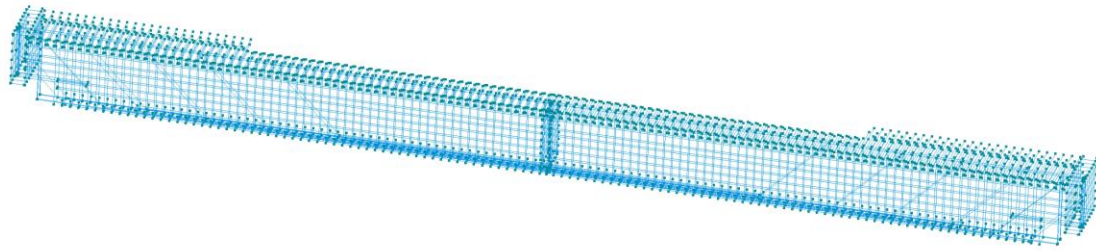


Figure 3.6. Imported lines into ATENA Pre, representing the ordinary reinforcement.

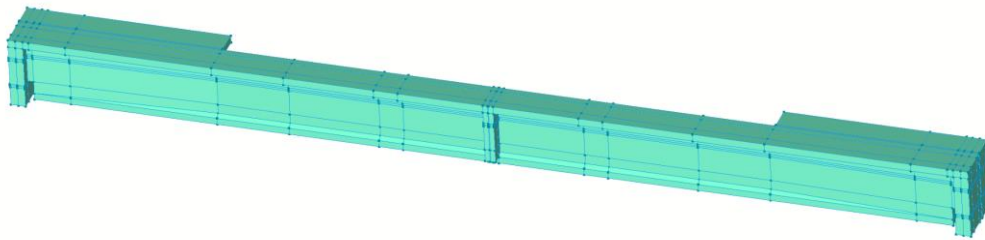


Figure 3.7. Model of the beam after the generation of the solid volumes in ATENA Pre.

3.5.2 Crossbeams and slab

The cast-in-place parts of the composite section were modelled as independent volumes: the 0.20 m slab, which forms the upper part of the section and extends mainly to one side because the element was an edge beam, and the crossbeams at the supports and at mid-span. Independent volumes are necessary because a different concrete will be assigned to the cast-in-place parts (Section 3.6.1). The slab was reproduced with the reduced width that resulted from the longitudinal cut performed before the test, when the element was separated from the adjacent deck [6].

Adjacent volumes were defined so that they share their boundary surfaces. The mesh is therefore continuous across the casting joint between the precast beam and the cast-in-place parts, which corresponds to assuming perfect bond at the joint. The assumption is supported by the behavior observed during the test, in which no slip of the slab with respect to the web

was reported [6]. It remains a simplification of the model: a real construction joint has a finite shear strength, and a refined model could introduce an interface there as well.

3.5.3 Supports and load distribution

The support region of the model reproduces the test configuration described in Chapter 2. Each end of the beam terminates with a Gerber saddle that rests, through a stack of steel plates and a neoprene sheet, on the vertical wall of a reinforced concrete plinth, called plinth A and plinth B [6]. In the model each plinth is a solid volume, the steel plates and the neoprene pad are thin solid volumes interposed between the saddle and the crossbeams, and the base of each plinth is the surface on which the fixed boundary conditions are applied (Section 3.10).

The live load of the test consisted of cast iron ballast stacked on the slab over the central band of the span. Each ballast layer was formed by four elements of 105 kN [6]. In the model, four loading surfaces were defined on the extrados of the slab, in the positions occupied by the four ballast elements of each layer. The load is applied on these surfaces as a distributed force in force control (Section 3.10). Finally, a thin solid volume was prepared at each end of the beam, on the side facing the SS28 state road, to represent the wooden shims inserted during the test as a lateral restraint. This volume is used by the parametric study of Section 3.10.4.

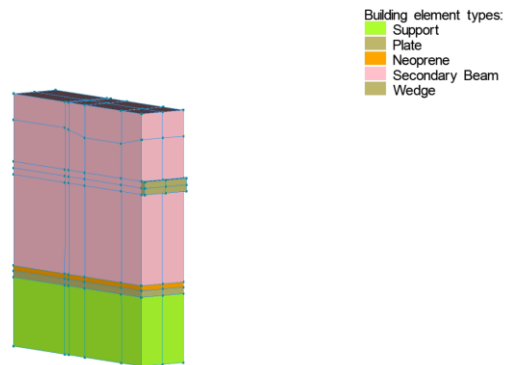


Figure 3.8. Scheme of the support region as reproduced in the model.

3.5.4 Ordinary reinforcement

All the ordinary reinforcement shown in the original drawings was reproduced in the model as discrete bars. The drawings report bars of 8, 10, 12, 22 and 28 mm diameter. The main groups are the twenty $\Phi 28$ longitudinal bars of the bottom chord, which are bent up towards the ends to act as shear reinforcement, the $\Phi 8$ stirrups of the web, the $\Phi 8$ transverse bars and the $\Phi 12$ longitudinal bars of the slab, and the further $\Phi 10$ and $\Phi 22$ bars present in the drawings. The bars were drawn in AutoCAD in dedicated layers, one for each group, and imported with the rest of the geometry.

Modelling the complete reinforcement, and not only the bottom bars, matters for two reasons. The stirrups and the slab bars control the shear behavior and the transverse distribution of the load, both of which are involved in the skew bending response of this asymmetric section. In addition, the comparison with the strain gauges placed on the $\Phi 28$ bars (Section 3.12) is meaningful only if the surrounding reinforcement is also present, since it influences the strain field after cracking.

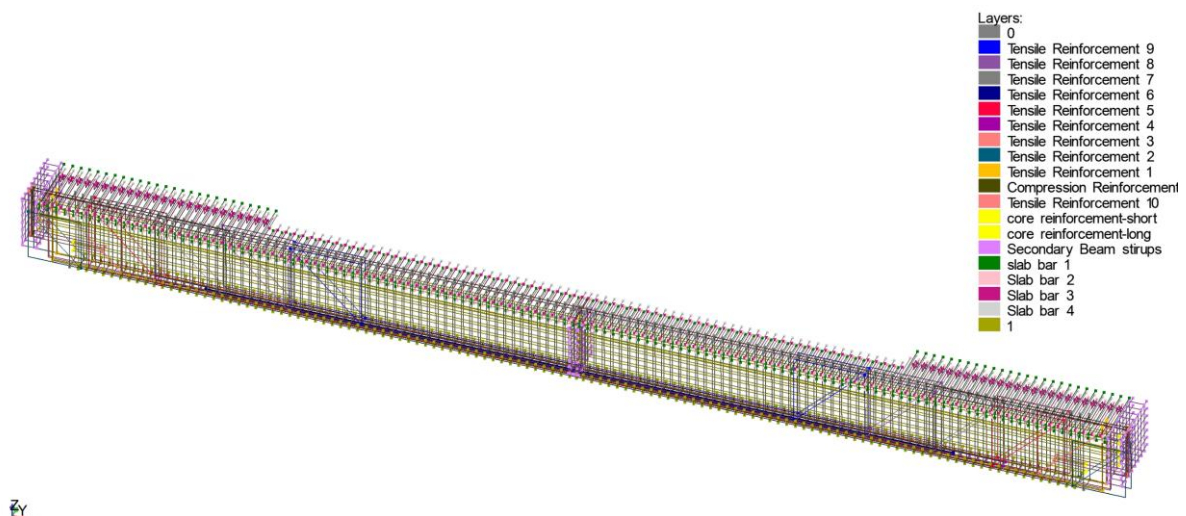


Figure 3.9. Model of the beam reinforcement and stirrups in dedicated layers.

3.6 Constitutive models of materials

This section presents the materials assigned to the model and the reasoning behind each choice. Since the purpose of the model is to reproduce a real test, mean material properties were

adopted, taken from the statistical evaluation of the destructive tests described in Chapter 2. Partial safety factors have no role at this stage; the safety formats discussed in 0 apply to assessment calculations, not to the validation of a model against an experiment.

3.6.1 Concrete

The phenomenon to be modelled is the full nonlinear response of concrete: cracking in tension, crushing in compression, the reduction of compressive strength caused by transverse cracking, and the partial closure of cracks during the unloading branches of the cyclic test. ATENA offers several concrete materials. I adopted the fracture-plastic prototype CC3DNonLinCementitious2, which is the default choice of the program and the standard model for static analyses of reinforced concrete [41]. It combines a smeared crack model with exponential tensile softening for tension and a plasticity surface of the Men trety-Willam type for compression [47] and it accounts for the biaxial interaction between the two regimes [46].

The parameters of each concrete were generated from the mean cylindrical compressive strength according to Model Code 2010 [18] using the dedicated option of the material dialogue. The program derives the tensile strength, the fracture energy and the complete stress-strain law from f_{cm} with the Model Code formulations. The laboratory campaign of Chapter 2 provided the compressive strengths of the cores, but no direct measurement of the elastic modulus, with one exception. I therefore kept the elastic moduli generated by Model Code 2010 for all the concretes, and replaced the generated value with the experimental one only for the web of the beam in the region where the collapse originated, the only zone for which a tested modulus is available. Four concrete materials were defined in this way, listed in Table 3.1: one for the precast beam, one for the cast-in-place slab and crossbeams, and two for the flange and the web of the beam in the collapse region, where the cores gave locally lower strengths.

Table 3.1. Concrete materials of the model: mean compressive strength and elastic modulus.

Concrete material	f_c [MPa]	E [GPa]	Source of E
Precast beam	46.05	35.770	MC2010 [18]
Slab and crossbeams	52.13	32.279	MC2010 [18]
Beam flange, collapse region	41.50	34.551	MC2010 [18]
Beam web, collapse region	43.80	29.626	Test [6]

Differentiating the collapse region is a deliberate choice. The statistical analysis of the cores in Chapter 2 showed that the strengths in that zone fall below the averages of the rest of the member. Assigning the local values where they were measured lets the model develop the failure where the real beam developed it, instead of forcing it through artificially uniform properties.

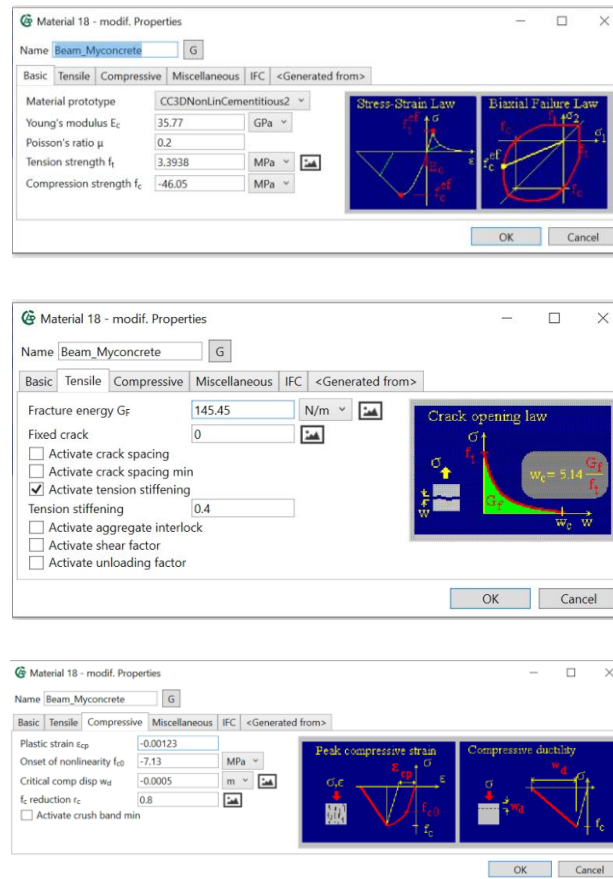
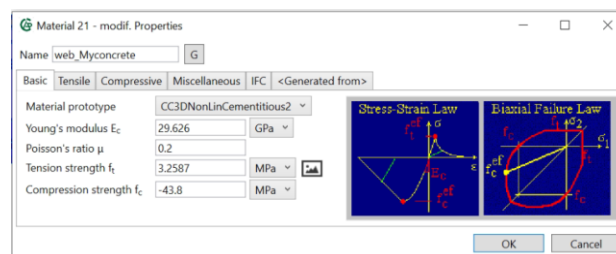


Figure 3.10. Material properties of the precast beam concrete.



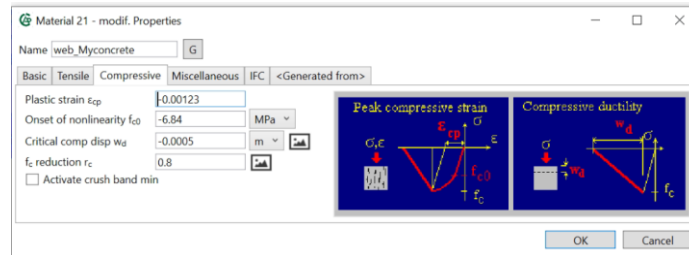
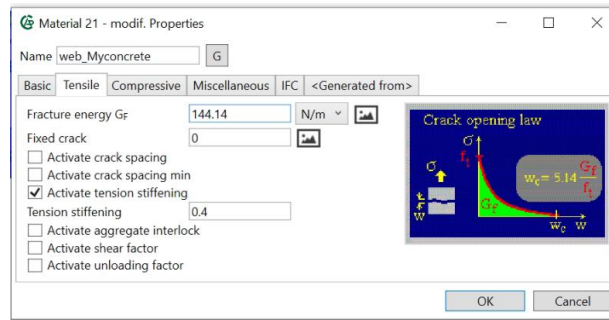
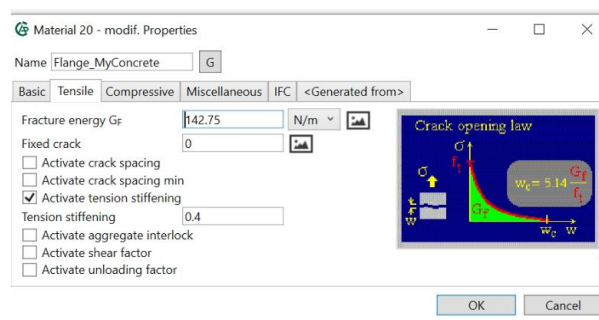
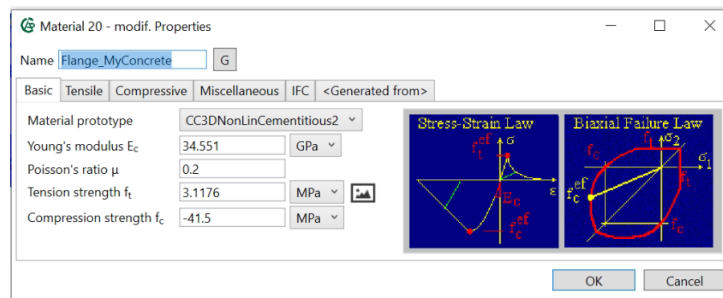


Figure 3.11. Material Properties of the concretes in web assigned to the collapse region.



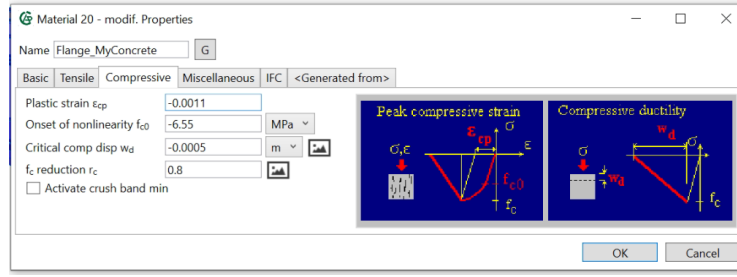


Figure 3.12. Material Properties of the concretes in top part of the collapse region.

Within the smeared crack framework, the software provides both fixed and rotating crack formulations [41]. In this work, the rotating crack model was adopted, in which the crack plane remains orthogonal to the principal tensile strain and continuously reorients when the principal direction evolves during the analysis. As a consequence, the crack stays coaxial with the principal strain, and shear stresses do not act across the crack plane, so no shear retention factor needs to be specified; this avoidance of an additional, hardly quantifiable parameter was the main motivation for the choice. In fixed crack formulations, the shear retention factor can induce stress locking and lead to an overestimation of stiffness when the principal stress direction rotates away from the initial crack orientation, whereas in the present test the load is always applied in the same direction and the loading–unloading cycles modify only the magnitude, not the sign, of the action. Under these conditions, the principal directions evolve without reversal, and the rotating crack model can follow them consistently throughout the cycles, keeping the constitutive description free from extra calibration parameters. In ATENA this behavior is obtained by setting the fixed crack coefficient equal to zero, so that no fixed crack effect is retained; the known limitation is that the crack orientation is updated at each step and does not preserve its direction after a strong reversal of the principal strain, a situation that is not relevant for the present non-reversing load history but would become important under fully reversed cyclic loading.

Further parameters of the concrete law deserve a justification. The tension stiffening factor c_{ts} represents the contribution of the concrete between cracks to the stiffness of the embedded reinforcement; it sets a lower bound to the tensile stress in the softening branch equal to $c_{ts} f_t$. I assumed $c_{ts} = 0.4$ for all the concretes, as recommended by the theory manual and consistent with Model Code 2010 [41], [18]. The choice is appropriate for the heavily reinforced bottom chord, where twenty $\Phi 28$ bars cross the tension zone. The compressive strength reduction due

to cracking parallel to the compression direction is applied by the program through the factor r_c , whose lower limit was kept at the default value of 0.8 [41].

3.6.2 Reinforcing steel

The reinforcement must reproduce yielding, hardening up to the ultimate strength, and the hysteretic response under the loading and unloading cycles of the test. The destructive campaign of Chapter 2 provided tensile tests on bar samples, whose statistical evaluation gives the mean properties of Table 3.2. Since the tested samples are the only experimental information available on the reinforcement, I assigned the same constitutive law to all the bar groups of the model, from the $\Phi 8$ stirrups to the $\Phi 28$ main bars. This is the most realistic option allowed by the available data; the alternative, namely assigning code-based nominal properties to the untested diameters, would mix measured and assumed values within the same member. The uniform assignment remains a limitation, because bars of different diameter and production batch may differ in strength, and it is recorded as such.

Table 3.2. Mean properties of the reinforcing steel from the tensile tests, assigned to all bar groups.

Property	Symbol	Mean value
Elastic modulus	E_s	200 GPa
Yield strength	f_y	397.2 MPa
Tensile strength	f_t	567.25 MPa
Ultimate strain	ϵ_u	5.81 %

The stress-strain law of each reinforcement material was introduced through a multilinear function built from the test results, and the materials were generated with the prototype CCCyclingReinforcement. This prototype adds the hysteretic rule of Menegotto and Pinto to the monotonic envelope, so the bars dissipate energy correctly during the load reversals of the cyclic test [44]. With a simple bilinear material without a cyclic rule, the unloading branches of the simulated load-displacement curves would be too stiff and the residual deflections of the test could not be reproduced.

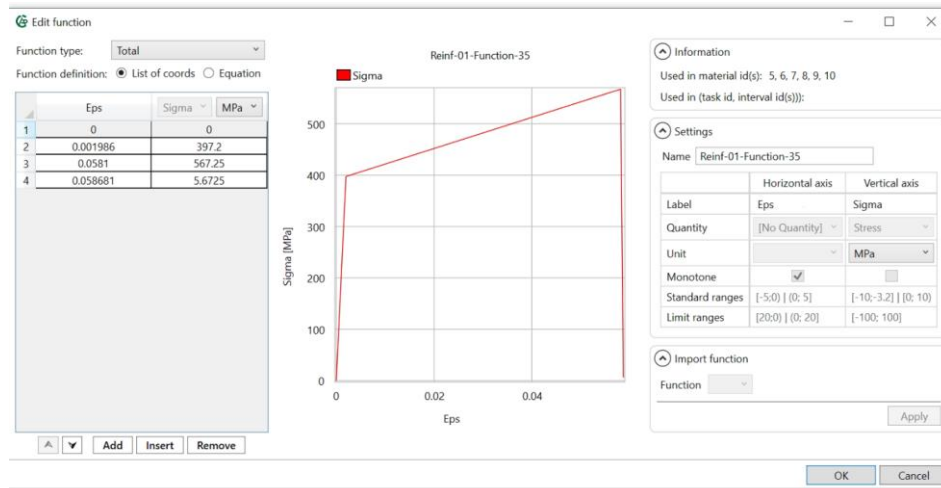


Figure 3.13. Definition of the reinforcement material from the experimental stress-strain function.

Perfect bond was assumed between every bar and the surrounding concrete. The bars of the beam are ribbed, no anchorage or splitting failure was observed during the test [6] and the comparison of interest concerns global displacements and bar strains at mid-span and at one third of the span, far from anchorage zones. Under these conditions a bond-slip law would add parameters that cannot be supported by measurements, without a clear benefit for the response quantities compared in Chapter 4. The assumption is a recognized simplification: it slightly overestimates the tension stiffening between cracks, an effect partially compensated by the calibrated tension stiffening factor of the concrete.

3.6.3 Elastic materials for plates and bearings

Three groups of auxiliary components are not expected to develop material nonlinearity and were modelled with linear elastic materials. The support plinths were assigned an elastic material with the modulus of concrete. Their own deformation does not belong to the response of interest, and the topographic measurements of Chapter 2 were already corrected for the settlements of the plinths [6] so an elastic plinth is sufficient. The steel plates of the support stack and the load distribution components were assigned an elastic steel with $E = 200$ GPa and Poisson ratio 0.3, so the load is spread realistically without introducing local plasticity that the test did not show.

The neoprene pad of the supports is the component that governs the support flexibility, and its stiffness cannot be settled with a single guess. In the baseline model it is a 3 cm elastic volume interposed between the saddle plates and the crossbeam. The definition of its stiffness, the

alternative materials tested for it, and its replacement with an interface element are the subject of the parametric study of Section 3.10, which is the core of the modelling work on the supports.

3.7 Creating and assigning structural elements

In ATENA the connection between the geometry and the materials is made through the structural element types, defined in the building element types section of the input tree. Each type couples a material with a mesh interpolation order and is then assigned to one or more geometric solids. The following types were created.

- Five beam-type elements with linear interpolation, assigned to the precast beam, to the crossbeams (the secondary beams of the model), to the support plinths and to the flange and web volumes of the collapse region, each with its concrete or elastic material from Section 3.6.
- Three plate-type elements with linear interpolation, assigned to the steel plates and to the volume that represents the wooden shims of the lateral restraint introduced from the third loading phase.
- One slab-type element with linear interpolation, assigned to the portion of the slab that remained after the longitudinal cut performed before the test.

All the solids use linear interpolation, consistent with the eight-node hexahedral elements of the mesh (Section 3.9). A further decision concerns the evolution of the materials along the analysis: each volume keeps a single material for the whole load history.

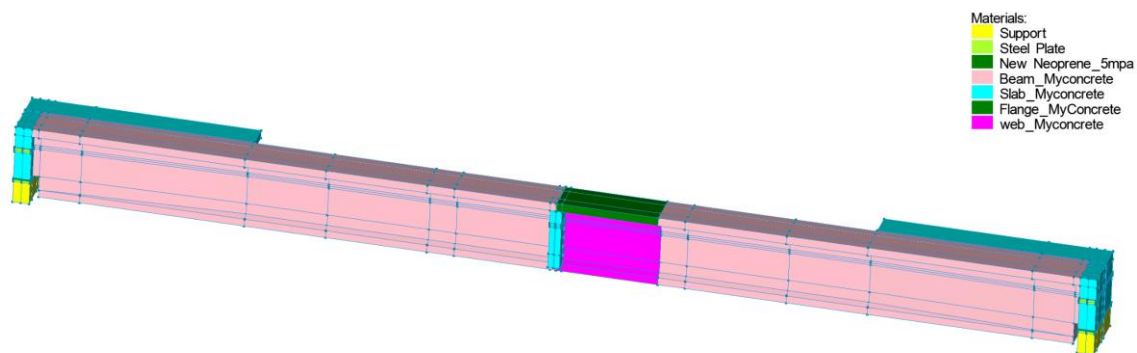


Figure 3.14. Definition and assignment of the structural element types.

3.8 Creating and assigning ordinary reinforcement

ATENA offers two ways to model the ordinary reinforcement: discrete bars, made of truss elements that carry only axial force, or smeared reinforcement, distributed in the concrete along up to three orthogonal directions [44]. I chose the discrete approach for all the bars. It is more expensive, because every bar has to be drawn, but it reproduces the real position of the reinforcement, which matters for the local comparison with the strain gauges placed on individual $\Phi 28$ bars.

Discrete bars in ATENA are embedded by default: the program cuts each bar at the boundaries of the finite elements it crosses and connects the resulting truss segments to the surrounding mesh. The reinforcement mesh is therefore independent of the solid mesh, and the bars do not constrain the meshing of the concrete [44]. Each truss segment is an isoparametric element with two nodes and one Gauss integration point.

For each of the six bar groups a reinforcement type was generated with the bar command, assigning the steel material of Section 3.6.2, the diameter of the group, from which the program computes area and perimeter, and perfect bond. Each type was then assigned to the corresponding linear entities, which arrive from AutoCAD already separated by layer. The layer discipline imposed during the drawing phase pays off here: the assignment of six reinforcement types to several hundred lines reduces to six selections by layer.

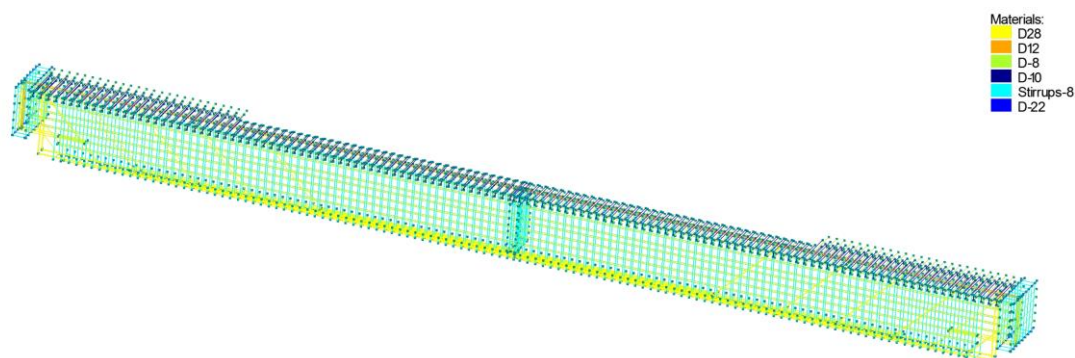


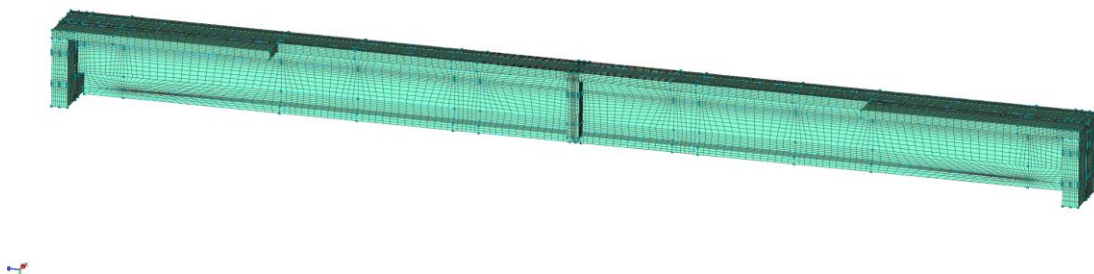
Figure 3.15. Reinforcement of the model after the assignment of the bar types.

3.9 Finite element mesh definition

The mesh is generated automatically by ATENA before the analysis, on the basis of the shape and size parameters entered in the FE mesh section of the input tree. Two decisions define it: the element shape and the element size.

For the shape, I selected a hexa-dominant structured mesh for all the solids. Eight-node hexahedral elements with linear interpolation are well suited to the prismatic volumes of this model, which were prepared in AutoCAD precisely with six-face solids in mind (Section 3.5). A structured hexahedral mesh gives regular element shapes, avoids the artificial stiffness scatter of distorted tetrahedra, and makes the crack band regularization of the concrete softening more uniform, since the element size in the direction of the principal stresses varies little across the model [48].

For the size, I assigned 0.20 m along the axis of the beam with the FE mesh size for line command. The choice balances accuracy and cost for a member of 25.90 m span: it places more than one hundred elements along the span and several elements through the height of the web, enough to describe the strain gradients of bending, while keeping the model solvable in acceptable times for the parametric study of Section 3.10. The objectivity of the softening response with respect to the element size is provided by the crack band approach of the concrete model, which scales the softening branch with the element dimension [41], [48]. A formal mesh sensitivity study was not performed at this stage because the model behaves regularly with the adopted size, and the changes in results were small with a variation of the mesh of 0.05m in longitudinal direction.



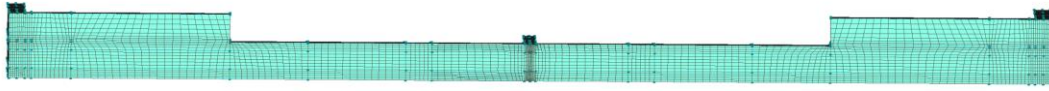


Figure 3.16. Finite element mesh of the model, global size 0.20 m along the beam axis.

The automatic generation produced a mesh of 86,983 nodes and 90,583 solid elements. The mesh is hexa-dominant. The 69,919 eight-node hexahedral elements form the regular core of the volumes prepared with six faces, and they make up about 77% of the solid elements. The remaining 18,881 four-node tetrahedra and 1,783 five-node pyramids are generated automatically in the transition regions, where the brick pattern cannot be mapped, mainly around the supports and at the junction between the cut slab and the web. Table 3.3 reports the composition and the shape quality of the mesh. The hexahedral core has a high mean quality of 0.93, with 96% of its elements in the best quality band. The tetrahedra and pyramids of the transition regions have lower mean values, 0.80 and 0.66. These lower-quality elements are a small fraction of the model. They are limited to the geometric transitions between non-aligned volumes.

Table 3.3. Composition and shape quality of the finite element mesh (86,983 nodes).

Element type	Number	Share
Hexahedra (8 nodes)	69,919	77%
Tetrahedra (4 nodes)	18,881	21%
Pyramids (5 nodes)	1,783	2%
Total	90,583	100%

3.10 Boundary conditions and modelling of the supports: the parametric study

The boundary conditions complete the mechanical model. They include all the constraints and actions applied to points, lines, surfaces and volumes, and ATENA organizes them in load cases, so that each group can be activated independently in the load history. The following load

cases were created in the analysis section of the input tree: Body force, for the self-weight; Supports, for the constraints at the plinths; Wedge support, for the lateral restraint introduced from the third loading phase; Ballast 1 to Ballast 4, for the loads of the four ballast positions.

The base constraints are simple. The lower surface of each plinth is fixed in the three directions, which reproduces the foundation of the test plinths; the measurements of Chapter 2 were corrected for the plinth settlements, so a rigid base is consistent with the data compared in Chapter 5 [6]. The self-weight is applied as a body load of 25 kN/m³ along the vertical axis to the volumes of the beam, crossbeams and slab. The live load is applied on the four loading surfaces of the slab extrados.

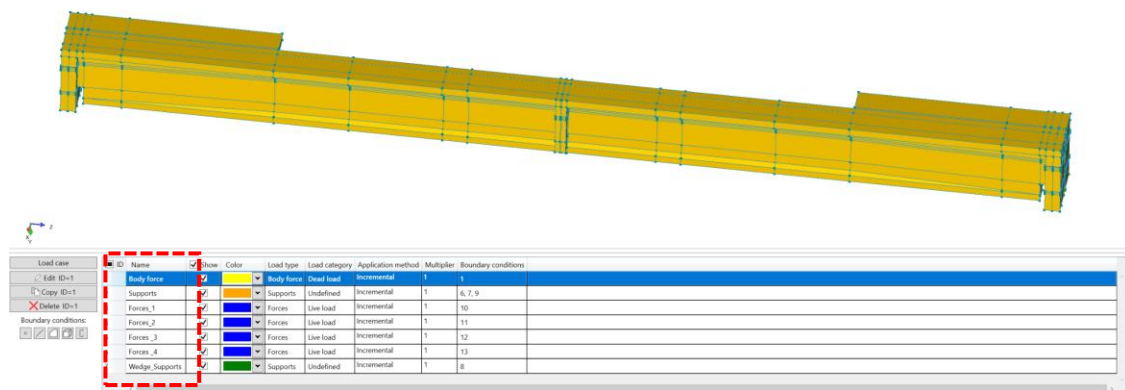


Figure 3.17. Load cases defined in the model.

ATENA lets the bearing volume be assigned different material models, and within each model the elastic modulus can be set freely. I first modelled the elastomeric pad with an elastic isotropic material and reduced its modulus in steps: $E = 10, 5$ and 3 MPa. The Poisson ratio was set to a value representative of the nearly incompressible elastomer. A single modulus controls both the vertical and the lateral response of an isotropic pad, so each value was judged against both displacement components measured at mid-span. I then repeated the three runs with a steel material at the same moduli of $10, 5$ and 3 MPa.

The second material was used to check whether the global response depends on the bearing stiffness alone, or also on the constitutive model assigned to the pad.

Each variant was run through the complete load history of Section 3.11 and compared with the measured vertical and transverse displacement at mid-span. The list of variants and the quantitative comparison belong to the calibration are reported in Chapter 4. The study confirms that the flexibility of the support is essential to reproduce the measurements. It also exposes a limitation of the approach: a single isotropic modulus couples the vertical and the lateral stiffness of the pad, so the two displacement components cannot be matched independently. This limitation, together with the inability of a bonded elastic volume to reproduce the relative movements at the contact, motivates the interface element of the next subsection.

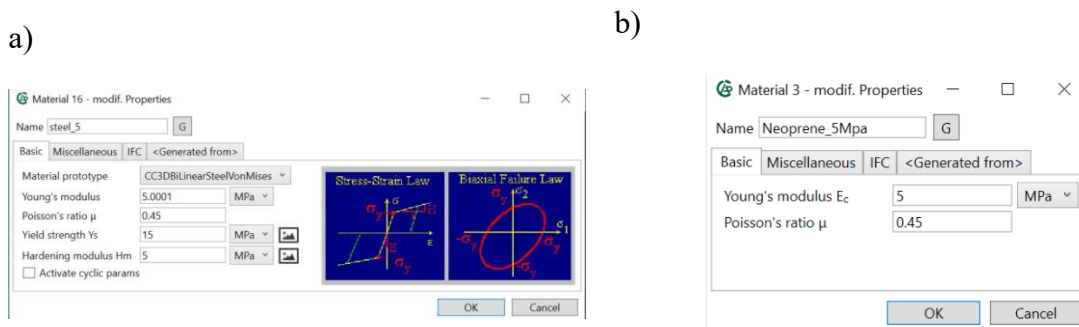


Figure 3.19. Definition of a) Steel and b) Elastic materials for bearing.

3.10.2 Interface element in place of the bearing

An elastic volume, however calibrated, keeps the elastomeric pad glued to the steel plate and to the crossbeam: it cannot open, and it transmits shear without limit. The test suggests otherwise. The relative displacements measured at the supports include a component that an attached elastic layer cannot supply, and the contact between rubber and concrete is frictional

in nature. The second step of the study therefore replaced the elastomeric pad volume with a contact element: the CCInterface material of ATENA, applied between the surface of the saddle plate stack and the crossbeams [44].

The interface follows the Mohr-Coulomb criterion. In the compression regime the shear strength grows with the normal stress:

$$|\tau| \leq c + \sigma_c \tan \Phi \quad (3.3)$$

where c is the cohesion, Φ the friction angle and σ_c the compressive stress acting on the interface; in tension the strength is cut off at the interface tensile strength f_t . Two families of parameters define the element. The first family (c, f_t, Φ) describes the physical strength of the contact, and the manual suggests for it a friction coefficient between 0.3 and 0.5 and small positive values of cohesion and tensile strength to avoid numerical problems [44]. The second family contains the normal and tangential stiffnesses K_{nn} and K_{tt} , each with a base and a minimum value, which govern the relative displacements before failure.

The treatment of the stiffnesses is the point where my approach differs from common practice. The manual, in the absence of data, recommends a penalty-type estimate, taking the base stiffness about ten times the stiffness of the adjacent finite elements, $K \approx 10 E/\Delta l$, a value that depends on the mesh and has no physical meaning [44]. In this model the interface replaces a real, measurable object: a rubber sheet of known thickness. Its stiffnesses can therefore be derived by physical equivalence with the layer it replaces. A layer of thickness t , modulus E and shear modulus G compressed between rigid faces responds with the stiffnesses per unit area:

$$K_{nn} = E / t \quad (3.4)$$

$$K_{tt} = G / t \quad (3.5)$$

The shear modulus follows from the elastic modulus through the Poisson ratio. Rubber is nearly incompressible, so I assumed $\nu = 0.5$, which gives:

$$G = E / [2(1 + \nu)] = E / 3 \quad (3.6)$$

Equations (3.4) to (3.6) tie the two stiffnesses together: for every elastic modulus tested in the parametric study, the corresponding shear modulus and the pair K_{nn} , K_{tt} are recomputed consistently, instead of being tuned independently. The procedure has two advantages over the penalty route. The stiffnesses are independent of the mesh, so refining the model does not change the support behavior; and each variant of the study keeps a clear physical identity, since it corresponds to one elastomer with one modulus. The minimum stiffnesses, which act only as numerical predictors when the interface opens, were set two to three orders of magnitude below the base values, within the range indicated by the manual [44].

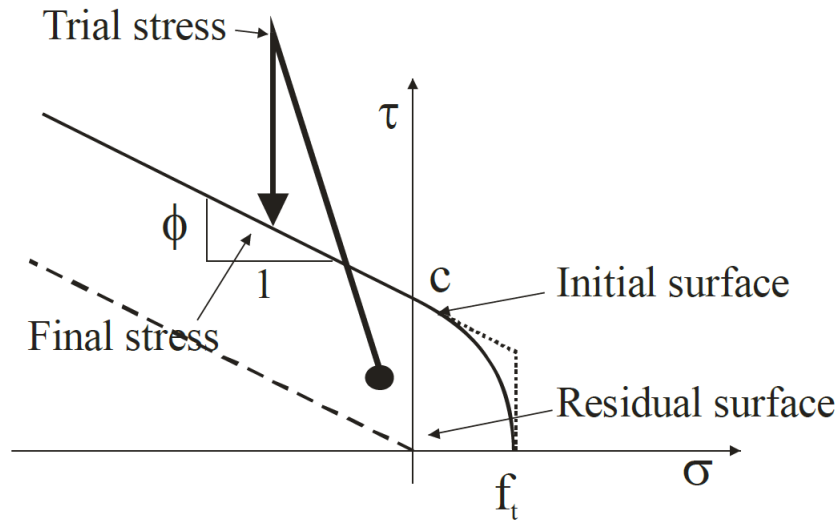


Figure 3.20. Failure criterion of the CCInterface element [44]

3.10.3 Steel plates modelled as elastic material with $E = 200$ GPa

Between the saddle and the elastomeric pad the test interposed a stack of steel plates [6]. In the model the stack is a solid with a linear elastic steel material, $E = 200$ GPa and Poisson ratio 0.3. The reasoning is one of contrast: the plates are far stiffer and far stronger than everything around them, so the deformation of the support region must concentrate where it physically occurs, in the elastomeric pad below and in the reinforced concrete of the saddle above. An elastic stack delivers exactly this behavior at no numerical cost. The limitation is the absence

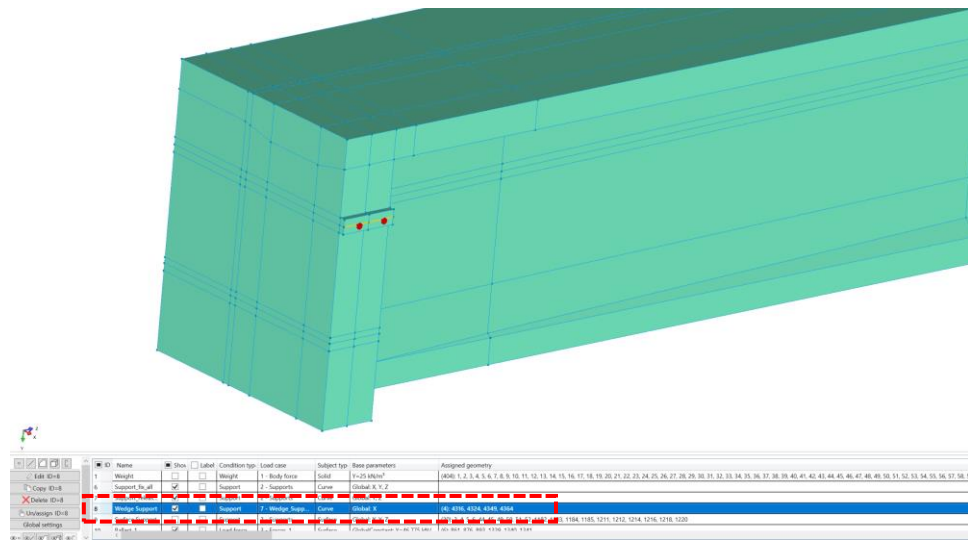
of local yielding and of contact nonlinearity between the individual plates; both effects are negligible at the stress levels of the test, and neither was observed during it [6].

3.10.4 Lateral restraint from the third cycle: spring stiffness study

After loading step 16 of the test, at the end of the second loading phase, wooden shims were inserted into the gap between the vertical wall of the plinths and the side of the beam facing the SS28 state road. The operation limited the torsional rotation of the beam, and the report shows that the transverse displacements decreased markedly in the following phases [6]. A model without this restraint may not follow the measurements beyond the second phase, so the shims must enter the model.

The restraint was introduced as a transverse condition on the end crossbeams, on the SS28 side, activated from interval 18 of the load history through a construction phase (Section 3.11). In the baseline it is a rigid line constraint in the transverse direction. A rigid constraint, however, overstates the shims: wood is deformable, the shims were inserted by hand into an irregular gap, and the contact works only in compression. The refinement of the study therefore replaced the rigid constraint with a point spring in the transverse direction and swept its stiffness over a range, from values approaching the rigid case down to soft values representing loose shims. The stiffness retained is the one that reproduces the measured reduction of the transverse displacement at mid-span after the insertion. A linear spring remains an idealization of a one-sided contact: it reacts in both directions, while the real shims can only push. The approximation is acceptable because, after the insertion, the beam keeps bearing against the shims for the rest of the test [6].

a)



b)

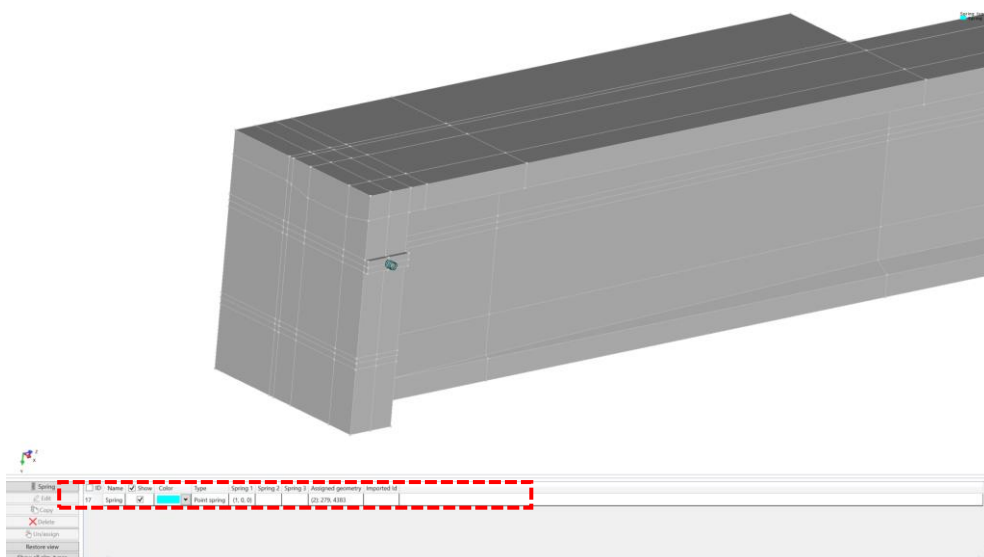


Figure 3.21. Lateral restraint of the test and its idealization in the model, a) fully restraint; b) additional transversal spring.

3.11 Load history

The load history assembles, in the order of the test, all the actions defined in the previous section: the support constraints, the self-weight, the lateral restraint and the loading and unloading forces of the ballast. In ATENA the history is built in the tasks section of the input

tree as a sequence of intervals, and each interval is divided into steps for a gradual application of its load. For every interval the program requires the active load cases, which remain active in the following intervals with the exception of the constraints, the solution method, the number of steps, the load multipliers, and the construction phases used to change the static scheme.

The experimental program of Chapter 2 consisted of four ballast layers, SA to SD, of 420 kN each, formed by four elements of 105 kN, applied in loading and unloading cycles [6]. To reproduce it, 49 intervals were defined, for a total of 106 steps. Interval 1 applies the support constraints and the self-weight in 10 steps. Each following interval applies or removes one ballast element of one layer, divided into 2 steps of 52.5 kN, so that the load grows gradually. The four ballast positions are loaded in alternation, which keeps the load arrangement symmetric at every instant and avoids spurious rotations from an unbalanced intermediate configuration. Intervals 2 to 17 cover the first two phases, two complete cycles on layer SA. At the beginning of interval 18 a construction phase activates the lateral restraint of Section 3.10.4, reproducing the insertion of the shims after step 16 of the test. Intervals 18 to 33 cover the third phase, a cycle up to layer SB and back to zero, and intervals 34 to 49 cover the fourth phase, the final loading through layers SA, SB, SC and SD.

The experimental beam collapsed at a total load of 1260 kN, during the completion of the third layer SC, at loading step 44 of the test [6]. In the model this load level corresponds to the end of interval 45. The history nevertheless defines the complete planned program up to 1680 kN, so that the analysis is free to proceed beyond the experimental collapse load and the resistance of the model can be read from the analysis itself, not imposed by the input. Table 3.4 reports the complete sequence.

Table 3.4. Load history of the model: 49 intervals and 106 steps reproducing the experimental loading program.

Interval	Steps	Phase	Action	Layer	Element	Total load [kN]
1	1–10	-	Self-weight and supports	-	-	-
2	11–12	1	Loading	SA	1	105
3	13–14	1	Loading	SA	2	210
4	15–16	1	Loading	SA	3	315
5	17–18	1	Loading	SA	4	420
6	19–20	1	Unloading	SA	1	315
7	21–22	1	Unloading	SA	2	210
8	23–24	1	Unloading	SA	3	105
9	25–26	1	Unloading	SA	4	0

Interval	Steps	Phase	Action	Layer	Element	Total load [kN]
10	27–28	2	Loading	SA	1	105
11	29–30	2	Loading	SA	2	210
12	31–32	2	Loading	SA	3	315
13	33–34	2	Loading	SA	4	420
14	35–36	2	Unloading	SA	1	315
15	37–38	2	Unloading	SA	2	210
16	39–40	2	Unloading	SA	3	105
17	41–42	2	Unloading	SA	4	0
18	43–44	3	Loading	SA	1	105
19	45–46	3	Loading	SA	2	210
20	47–48	3	Loading	SA	3	315
21	49–50	3	Loading	SA	4	420
22	51–52	3	Loading	SB	1	525
23	53–54	3	Loading	SB	2	630
24	55–56	3	Loading	SB	3	735
25	57–58	3	Loading	SB	4	840
26	59–60	3	Unloading	SB	1	735
27	61–62	3	Unloading	SB	2	630
28	63–64	3	Unloading	SB	3	525
29	65–66	3	Unloading	SB	4	420
30	67–68	3	Unloading	SA	1	315
31	69–70	3	Unloading	SA	2	210
32	71–72	3	Unloading	SA	3	105
33	73–74	3	Unloading	SA	4	0
34	75–76	4	Loading	SA	1	105
35	77–78	4	Loading	SA	2	210
36	79–80	4	Loading	SA	3	315
37	81–82	4	Loading	SA	4	420
38	83–84	4	Loading	SB	1	525
39	85–86	4	Loading	SB	2	630
40	87–88	4	Loading	SB	3	735
41	89–90	4	Loading	SB	4	840
42	91–92	4	Loading	SC	1	945
43	93–94	4	Loading	SC	2	1050
44	95–96	4	Loading	SC	3	1155
45	97–98	4	Loading	SC	4	1260
46	99–100	4	Loading	SD	1	1365
47	101–102	4	Loading	SD	2	1470
48	103–104	4	Loading	SD	3	1575
49	105–106	4	Loading	SD	4	1680

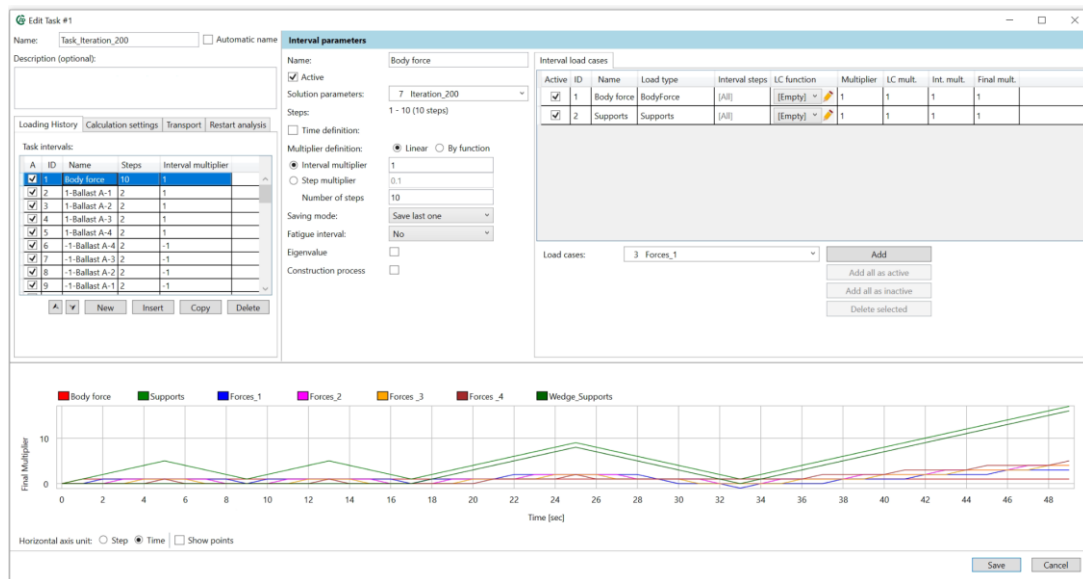


Figure 3.22. First interval of the load history.

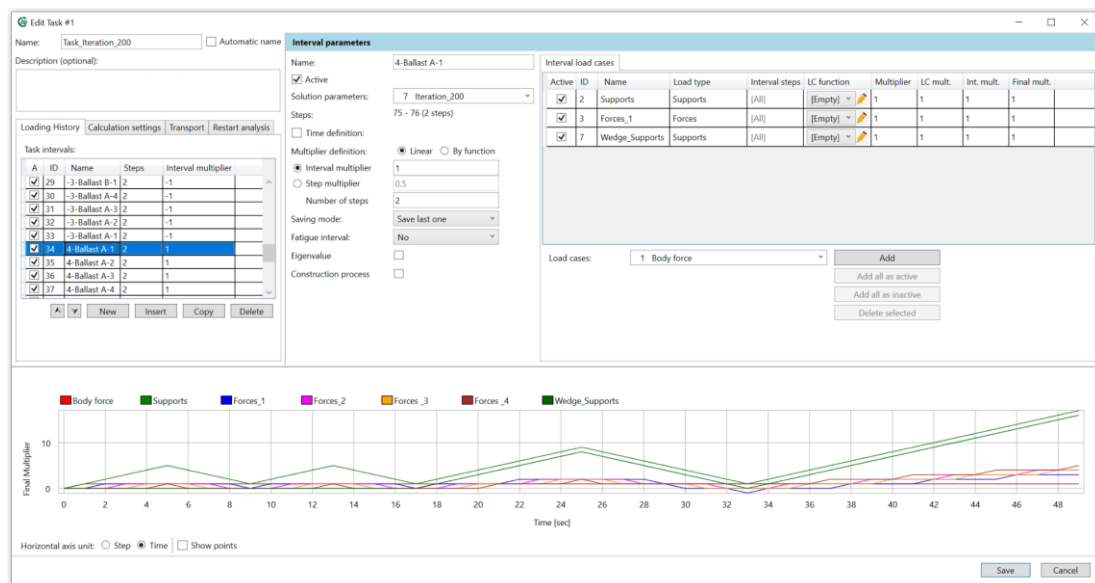


Figure 3.23. Typical loading interval of the third and fourth phases.

3.12 Monitoring points

Monitors are the quantities extracted from the analysis at chosen positions, step by step. They were placed at the positions of the instruments of Chapter 2, so that the numerical and the

experimental histories can be compared directly in Chapter 4 and , and ATENA exports them in tabular form after every step.

Before listing the monitors, it is useful to recall how the program produces results away from the integration points, because the choice of the monitor type depends on it [44]. The quantities are first evaluated at the integration points, where the equations are actually solved. They are then projected to the nodes of each finite element by the smoothing procedure of Zienkiewicz, and finally averaged at the nodes of the geometric model, with weights proportional to the element volumes. Nodal quantities, such as displacements, reactions and external forces, are best monitored at nodes; material quantities, such as stresses and strains, are best monitored at the nearest integration point, where they are computed without post-processing. The monitor type Monitor for point nearest node or IP applies this rule automatically.

The following monitors were defined in the model.

- The displacements of the points corresponding to the topographic targets of the P series of Chapter 2, in the longitudinal, transverse and vertical directions. Among them, P10 at mid-span at slab level and P09 at mid-span on the bottom chord are the primary monitors of the comparison: their vertical difference describes the deformation of the full section height at mid-span, and their transverse components describe the skew response of the beam.
- The axial strain of the reinforcement at the positions of the two strain gauges of the test: SGSX, on a $\Phi 28$ bottom bar at one third of the span, and SGMZ, on a $\Phi 28$ bottom bar at mid-span. These monitors are attached to the reinforcement, not to the concrete, and read the bar strain at the nearest integration point of the embedded truss.
- The longitudinal strains at the positions of the LVDT alignments of Chapter 2.
- The integrals of the vertical reactions on the fixed surfaces of the two plinths, and the total applied force of the ballast load cases. Together they provide the load axis of every load-displacement curve and a permanent check of global equilibrium.

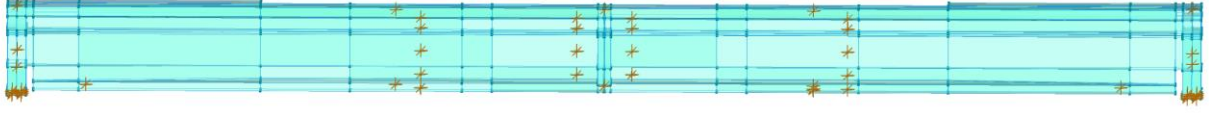


Figure 3.24. Monitoring points defined in the model.

3.13 Solution method and convergence criteria

The system of nonlinear equilibrium equations (3.2) is solved with the Newton-Raphson method. The choice follows from the nature of the simulation. The test is force controlled, and the load values of Table 3.4 must be met exactly at the end of every interval, since they are the weights of the ballast; the Newton-Raphson method keeps the load increment constant and iterates on the displacements, which is precisely this situation [41]. The alternative offered by the program, the arc-length method, iterates on loads and displacements together and is the right tool near limit points and along post-peak branches, but it does not guarantee exact load levels and is unsuitable for loads of fixed intensity such as the self-weight. Since the aim of the analysis is to trace the response up to the peak along the prescribed force history, and not to follow the descending branch, the Newton-Raphson method is the consistent choice.

At each iteration the method solves:

$$K(p_{i-1}) \Delta p_i = q - f(p_{i-1}) \quad (3.7)$$

$$p_i = p_{i-1} + \Delta p_i \quad (3.8)$$

where p is the vector of the nodal displacements, q the vector of the applied nodal forces, $f(p)$ the vector of the internal nodal forces, $K(p)$ the stiffness matrix and i the iteration counter. The program allows the stiffness matrix to be kept elastic, recomputed at each step, or recomputed at each iteration. I selected the tangent stiffness recomputed at each iteration, the full Newton-Raphson scheme. The model is strongly nonlinear, with cracking that opens and closes under

the load cycles, and the tangent scheme converges in fewer iterations near the heavily damaged states, at the cost of one matrix assembly per iteration. The line search option was also activated: it scales the displacement increment within each iteration, accelerating or damping the correction, and stabilizes the convergence in the steps where new cracks form [41].

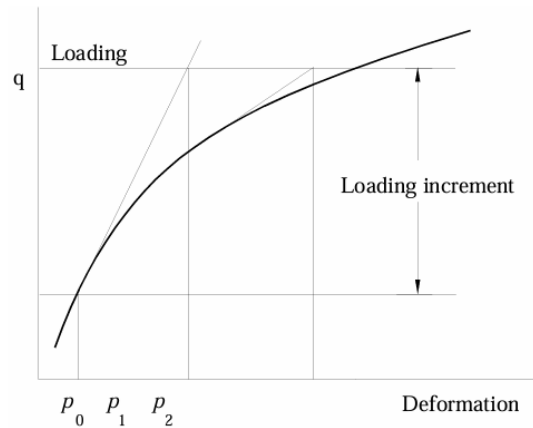


Figure 3.25. Scheme of the Newton-Raphson method [41].

The iterations of a step stop when four convergence criteria are satisfied at the same time [41]:

$$\|\Delta p_i\| / \|p_i\| \leq \varepsilon_{rel.disp} \quad (3.9)$$

$$\|q - f(p_{i-1})\| / \|f(p_i)\| \leq \varepsilon_{rel,force} \quad (3.10)$$

$$|\Delta p_i^T (q - f(p_{i-l}))| / |p_i^T f(p_i)| \leq \varepsilon_{rel.energy} \quad (3.11)$$

$$\max |q_k - f_k(p_{i-l})| / \max |f_k(p_i)| \leq \varepsilon_{abs,force} \quad (3.12)$$

The first criterion bounds the norm of the displacement increments, the second the norm of the unbalanced forces, the third the unbalanced work, and the fourth, where k denotes the single component, the maximum component of the unbalanced forces. The tolerances were kept at the default values of the program, listed in Table 3.5, and the default iteration limit per step was retained. The conditional break multipliers were set to 10 for the first, second and fourth criteria and to 1000 for the energy criterion, so that the analysis stops if the residual error at the end of a step exceeds ten times the tolerance; relaxing the tolerances instead of stopping would

silently corrupt the results. The equation solver was kept at the default PARDISO option, a direct sparse solver that is faster than the LU alternative and proved robust for this model.

Table 3.5 .Convergence tolerances adopted in the analyses (program defaults).

Criterion	Symbol	Tolerance
Relative displacement error	$\epsilon_{\text{rel.disp}}$	0.01
Relative force error	$\epsilon_{\text{rel.force}}$	0.01
Relative energy error	$\epsilon_{\text{rel.energy}}$	0.0001
Absolute force error	$\epsilon_{\text{abs.force}}$	0.01

With the solution parameters of this section the model is complete: geometry, materials, elements, reinforcement, mesh, boundary conditions, load history and monitors reproduce the test of Chapter 2 in every aspect that the available data can support. The calibration of the open parameters identified in Section 3.10 and the comparison between the numerical and the experimental response are the subject of Chapter 5.

Chapter 4. Model variants and calibration: the parametric study

4.1 Introduction and approach

The model described in the previous chapter fixes the beam geometry, the reinforcement, the four concrete zones, the mesh, the load history, and the solver. One region was left open for investigation. The support zone at the Gerber saddles governs how the beam settles and rotates, and the measured displacements depend directly on its behavior. I therefore treated the support as the subject of a parametric study.

The aim was to calibrate the support against the measured response through finite element model updating [49]. Every other feature of the model stayed frozen. Only two features were allowed to vary: the representation of the elastomeric bearing pad, and the way the lateral restraint from the wooden shims enters in the third loading phase. Any change in the computed response then follows from the support alone.

The study proceeded in three stages. The first stage asked which material model and which stiffness for the bearing reproduce the measured movements. The second stage replaced the bearing volume with a contact interface, to test whether frictional contact describes the support better than a bonded layer. The third stage examined whether the wooden shims act as a rigid block or as a deformable spring.

The comparison criterion stayed the same throughout. I compared the numerical histories of vertical and transverse displacement at mid-span, recorded at monitor P10, against the experimental curves from the test report [6]. The closeness of these histories, phase by phase, decided which configuration to retain.

4.2 Stage 1: the bearing as a physical volume

The elastomeric pad is the softest element in the load path at the support. Its stiffness controls how far the saddle settles and, through that settlement, the displacement at mid-span. The in-situ stiffness of an aged rubber pad is uncertain, so I treated its modulus as the parameter to determine rather than a fixed input.

Two questions had to be separated. The first was the value of the bearing stiffness. The second was whether the global response depends on stiffness alone, or also on the constitutive model used for the pad. To address both, I tested the pad with two material laws and three moduli. The laws were a linear elastic isotropic model and a bilinear steel model. The moduli were 10, 5, and 3 MPa, which bracket the plausible range for the compressed pad. The bilinear steel law stays elastic at these stress levels, with a Poisson's ratio of 0.3. The elastic isotropic law uses a near-incompressible Poisson's ratio close to 0.5, as expected for rubber.

All six variants were run through the complete load history, with the lateral restraint held at a rigid line constraint. The bilinear steel pad matched the measured displacements more closely than the elastic isotropic pad at the same modulus. Within each law, the value of 5 MPa gave the best agreement. The decisive difference between the two laws was the Poisson's ratio.

The near-incompressible pad resists spreading under vertical load. It couples the vertical and the lateral stiffness, so a single modulus must satisfy both displacement components at once, and the fit suffers. The steel pad, with its lower Poisson's ratio, compresses and spreads more freely. This decouples the two directions and tracks the measurements more accurately. I therefore carried the bilinear steel pad at 5 MPa forward as the baseline bearing for the next stages.

4.3 Stage 2: the bearing replaced by an interface

A bonded volume cannot open under tension and transmits shear without limit. The measured support displacements suggest relative movement between the rubber and the concrete that a bonded layer cannot reproduce. For this reason, I replaced the elastometric pad volume with a contact interface element of type `CCInterface`, placed between the steel plate stack of the saddle and the crossbeam. Its behavior follows the Mohr-Coulomb criterion described in Section 3.10.2.

The interface stiffnesses were not taken from a penalty rule. I derived the normal and tangential stiffnesses by physical equivalence to the rubber layer, using equations (3.4) to (3.6). The cohesion and the tensile strength were kept at small calibrated values, consistent with the support displacements, and the friction angle was held within the range recommended by the manual [44]. The first interface run used an equivalent modulus of 10 MPa, from the upper end of the Stage 1 sweep, and kept the rigid transverse restraint of the earlier stage.

This change marked the turning point of the study. Through the first two loading phases the model tracked the experimental curves closely, for both the vertical and the transverse displacement at mid-span, and reproduced the loading and unloading cycles.

The agreement broke down in the third loading phase. Once the lateral restraint activated, the model became too stiff in the transverse direction, and the computed displacement fell below the measured values. The cause was physical rather than numerical. The wooden shims were hand-fitted pieces in an irregular gap, able to act only in compression, so they could not have provided the rigid restraint the model imposed. The transverse displacement therefore diverged from the measurements as the load increased.

4.4 Stage 3: the lateral restraint as a spring

The result of Stage 2 pointed at the restraint, not the interface. I replaced the rigid transverse line with a transverse point spring on the end crossbeams, on the SS28 side, acting from the first loading phase. The spring stiffness was swept from values close to rigid down to soft values that represent loose shims. This let the shim stiffness, rather than an assumed rigid block, set the restraint.

A starting stiffness of 100 MPa was applied to both bearings. It improved the third-phase response against the rigid case, but the section was still too constrained, and the measured transverse displacement exceeded the computed value. As the spring was softened, the two bearing lines behaved differently.

For the model in which elastomeric pads are modelled as a volume with steel material, softening the spring improved the match, and removing the spring altogether gave the best result. With no transverse spring, the model still reproduced the measured transverse displacement in the third and fourth phases, through the flexibility of the pad alone. Any spring, even a soft one, over-constrained the section. The best steel configuration is therefore the bilinear pad at 5 MPa with no lateral spring, with the shim effect represented implicitly through the calibrated pad stiffness.

For the interface, the spring could not be removed. Deleting it caused the analysis to fail, because the section lost its transverse stability and the solution became unstable before reaching collapse. The spring was therefore retained. To keep the overall transverse stiffness balanced, I reduced the equivalent modulus and the spring stiffness together. Three paired values gave

the closest agreement: a modulus of 8 with a spring of 20, a modulus of 5 with a spring of 15, and a modulus of 3.5 with a spring of 12. The softest pair, a modulus of 3.5 MPa with a spring of 12, matched the measurements best.

4.5 The two final models

The study produced two final models, one for each representation of the bearing. The full programmed is summarized in Table 4.1, and the two retained models are compared in Table 4.2.

Table 4.1. Configurations tested in the parametric study.

Stage	Bearing model	E [MPa]	Lateral restraint	Activation phase	Spring, kt(MPa)
1	Elastic isotropic volume	10, 5, 3	Rigid line	Phase 3	None
1	Bilinear steel volume	10, 5, 3	Rigid line	Phase 3	None
2	Mohr-Coulomb interface	10	Rigid line	Phase 3	None
3	Bilinear steel volume	5	Transverse spring	Phase 1	100
3	Bilinear steel volume	5	None	None	None
3	Mohr-Coulomb interface	10	Transverse spring	Phase 1	100
3	Mohr-Coulomb interface	8	Transverse spring	Phase 1	20
3	Mohr-Coulomb interface	5	Transverse spring	Phase 1	15
3	Mohr-Coulomb interface	3.5	Transverse spring	Phase 1	12

Table 4.2. Parameters of the two final models.

Feature	Model 1 (steel bearing)	Model 2 (interface bearing)
Bearing representation	Bilinear steel volume	Mohr-Coulomb interface
Equivalent modulus	5 MPa	3.5 MPa
Poisson's ratio	0.3	0.5 (gives $G = E / 3$)
Lateral restraint (third phase)	None	Transverse spring, $kt = 12$ MPa

Model 1 uses the bilinear steel pad as a deformable volume, at an elastic modulus of 5 MPa, with no lateral spring. Model 2 replaces the pad with a Mohr-Coulomb interface, at an equivalent modulus of 3.5 MPa, with a transverse spring of stiffness 12 kept active for the entire analysis.

Both models keep every other feature of the analysis from the previous chapter. Both reproduce the global response of the test: the stiffness of the early cycles, the reduction in transverse displacement after the shims were inserted, and the loss of convergence near the experimental collapse load of 1260 kN. They differ in how they carry the support. The steel pad is a single deformable volume. It is simple and numerically stable, and it absorbs the shim effect into its own stiffness. The interface is a frictional contact that can open and slip. It is closer to the physical support, but it needs an explicit transverse spring for both accuracy and stability.

I retained both models on purpose, rather than reducing the work to one. Where the two agree with each other and with the test, the result is robust. Where they differ, the difference shows how sensitive the response is to the way the support is modelled. The detailed comparison of the deformed shapes, the failure modes, the displacements at the monitored targets and the bar strains is presented in the next chapter.

Chapter 5. Comparison of the final models with the experiment, and conclusions

The parametric study of Chapter 4 converged on two models that reproduce the measured response of the beam with comparable accuracy, and these two models are carried into the present chapter for a detailed comparison with the experiment. In the first model the bearing is retained as a physical volume and is assigned as a steel material with an equivalent modulus of 5 MPa, without any additional transversal restraint. In the second model the bearing volume is replaced by a Mohr-Coulomb interface with an equivalent modulus of 3.5 MPa, to which a linear transverse spring of 12 MPa is added and kept active for the entire analysis. Their configuration is summarized in Table 5.1.

The comparison is developed quantity by quantity. It begins with the deformed shape and the failure mode, proceeds through the displacements recorded at the monitoring targets, the rotation of the cross-section and the reinforcement strains measured by the strain gauges, and concludes with the general findings of the thesis. The experimental quantities are taken from the test report [6] whereas the numerical quantities correspond to the monitor histories defined in Section 3.12 and exported step by step after neutralizing the effect of self-weight. Each comparison is presented against the loading step and, where it clarifies the structural response, against the total applied load, so that both the deformation history and the apparent stiffness of the beam can be examined in the same plot.

Table 5.1. The two final models carried forward from Chapter 4.

Model	Bearing model	Equivalent modulus E [MPa]	Transverse spring
Model 1, steel bearing	Steel, bilinear	5	None
Model 2, interface bearing	CCInterface, Mohr-Coulomb	3.5	Linear, 12 MPa

In every other respect the two models are identical and follow the definitions of Chapter 3, namely the geometry, the four concrete zones of Table 3.1, the single constitutive law assigned to the reinforcement, the mesh of global size 0.20 m, the load history of 49 intervals and the

Newton-Raphson solution scheme. The only differences between them concern the bearing and the additional transversal spring. The comparison therefore isolates the influence of these two modelling choices, because every remaining source of difference is held constant.

5.1 Deformed shape and failure mode

The beam reached collapse during the placement of the third ballast layer SC, at a total applied load of 1260 kN [6]. Failure occurred under increasing load rather than on an unloading branch, and the beam collapsed at a load below the resistance expected from the historical design data [6]. Because the cross-section is that of an edge girder and is therefore strongly asymmetric, the response of the beam was never purely flexural: bending and torsion developed simultaneously, and the configuration at maximum deflection combined a pronounced vertical sag at mid-span with a transverse displacement of the upper slab.

A further observation emerged once the beam had failed and could be inspected. The reinforcement of bottom chord consisted of 19 bars of 28 mm diameter rather than the 20 reported in the design documentation, and I therefore built both models with 19 tensile bars at bottom chord of the beam, so that the reinforcement represented in the analysis corresponds to that actually present in the member. The absence of one bar reduces the flexural capacity of the section and contributes to explaining why the measured collapse load lies below the value predicted from the nominal design data.

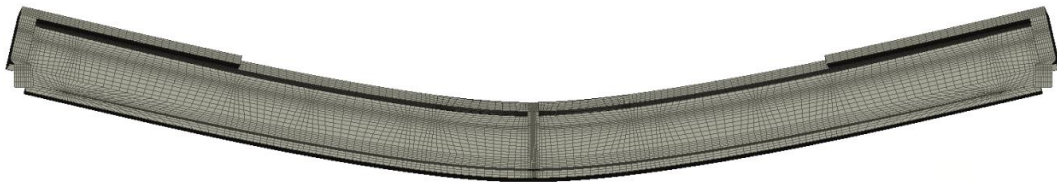
The model was also configured so that failure could develop freely at the location observed in the test. The statistical evaluation of the extracted cores indicated locally lower strengths in the web of the beam at mid-span, and this region was therefore assigned its own concrete materials, as reported in Table 3.1. The reduced web strength directs the limit state towards the collapse region without prescribing it explicitly, since the location of failure follows from the combination of the weaker material and the bending moment envelope.

Figure 5.1 compares the deformed configuration at maximum deflection for three cases, namely the real beam at collapse, the interface model and the steel model, each taken at the step of largest deflection. The deflected shape is essentially the same in the three cases: the vertical sag is concentrated at mid-span, and the upper slab undergoes a transverse displacement of the same sign and of comparable form. The choice of bearing model does not alter the deformation pattern but only the magnitude of its components, which are quantified in Section 5.2, so that the shape of the deflected beam is preserved across both numerical models.

a)



b)



c)



Figure 5.1. deformed shape at maximum deflection. (a) real beam at collapse; (b) steel model; (c) interface model.

I show the same configuration from two viewpoints, so that its two components can be read separately. The side view in Figure 5.2 isolates the vertical sag, which concentrates at mid-span while the support regions stay close to their initial position. The transverse view in Figure 5.3 shows the torsional rotation of the section together with the lateral displacement of the upper slab. In both views the steel model and the interface model give the same shape, which confirms that the bearing model governs the size of the movement and not its form.

a)

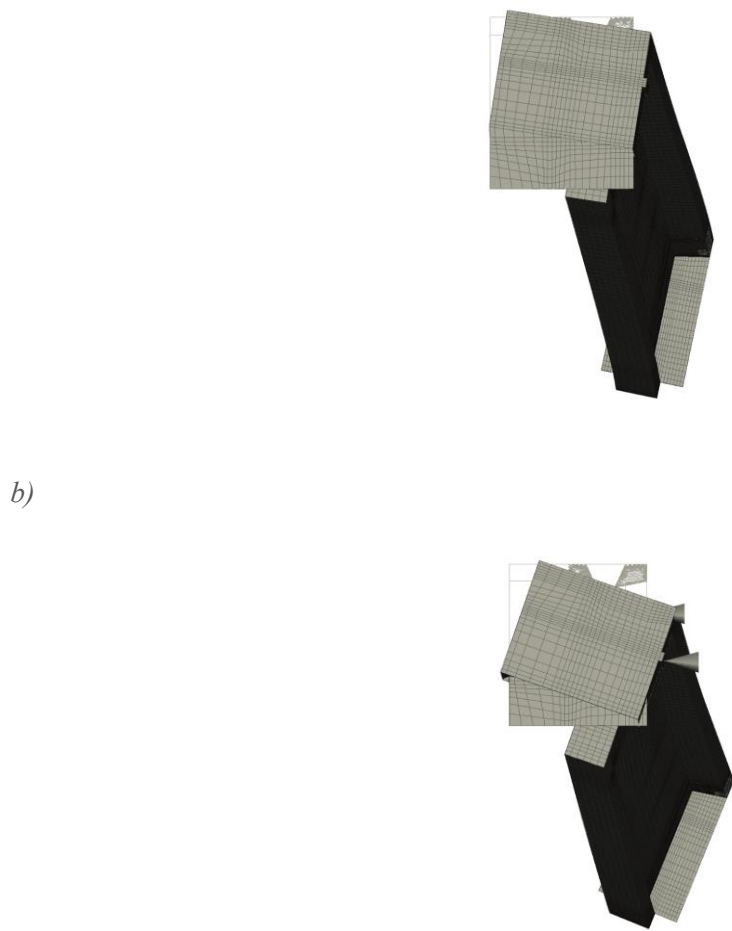


Figure 5.2. Deformed shape at maximum deflection, side view. (a) steel model; (b) interface model.



Figure 5.3. Deformed shape at maximum deflection, transverse view. (a) steel model; (b) interface model.

Figure 5.4 presents the crack pattern obtained from the two models at the same load level. Two families of cracks can be distinguished, and both are consistent with the expected behavior of

a beam of this type. Inclined cracks oriented at approximately 45 degrees develop near the supports, where the shear force is largest, and correspond to shear cracking, whereas vertical flexural cracks develop at mid-span, where the bending moment is largest, accompanied by longitudinal openings along the soffit and the web in the same region. Within the rotating crack model each crack forms normal to the principal tensile direction and reorients as this direction rotates under increasing load, which explains why the inclined cracks near the supports and the vertical cracks at mid-span are reproduced by the same model without any crack direction being imposed a priori.

a)



b)

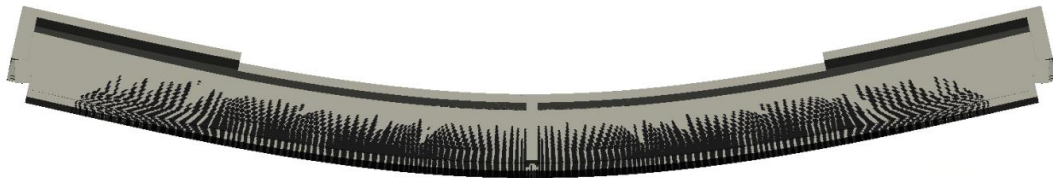


Figure 5.4. crack pattern of a) the steel model and b) the interface model at the load step near collapse. The 45-degree shear cracks near the supports and the bending cracks at mid-span.

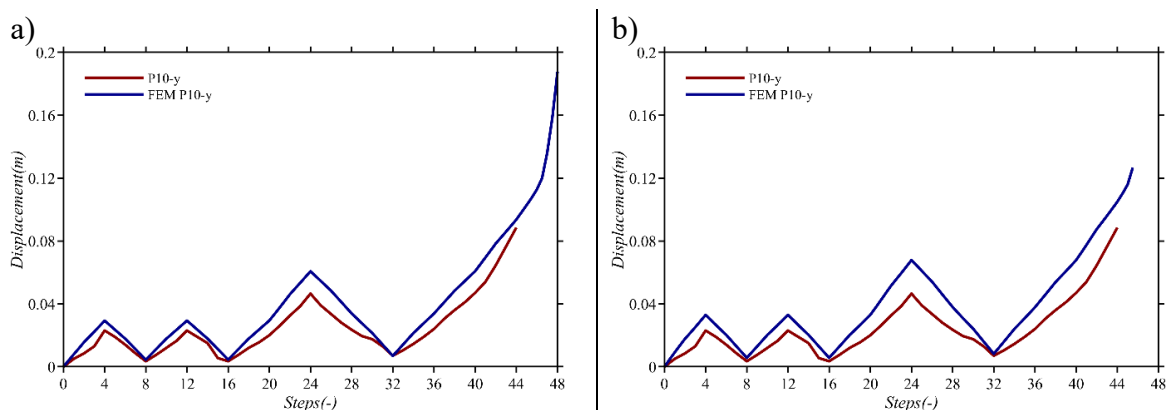
Both analyses attain their limit at a total load close to the experimental value of 1260 kN. The collapse load of each model is obtained from the analysis itself and is not imposed through the input, since the load history defines the complete planned programme up to 1680 kN, as described in Section 3.11. The agreement on the collapse load and on the failure region confirms that the two bearing models do not distort the capacity of the beam; rather, they

govern the evolution of the displacements, while the resistance is dictated by the flexural mechanism developing in the collapse region.

5.2 Displacement at the topographic targets

The displacements were monitored at the points that reproduce the topographic targets of the test, as defined in Section 3.12. The comparison at mid-span is carried mainly by two of these targets: P10, located at the level of the slab, and P9, located on the bottom chord of the same section. The vertical and transverse movements of these two points describe respectively the deflection and the transverse skew of the section, which are the quantities most directly controlled by the bearing. Each component is compared with the experiment in two complementary forms, against the loading step and against the total applied load, so that both the deformation history and the apparent stiffness of the beam can be examined.

Figure 5.5 compares the vertical displacement of P9 and P10 with the experimental record. When plotted against the loading step, both models follow the measured cyclic history, reproducing the loading and unloading branches as well as the residual displacement on return to zero, and the steel model in particular remains very close to the measurement throughout the test. When the same response is plotted against the total applied load, the slope of the curve represents the apparent vertical stiffness of the beam; in both models this slope is somewhat lower than that of the real beam, so that the numerical beam is slightly more flexible in bending and develops a larger deflection towards the end of the loading programme.



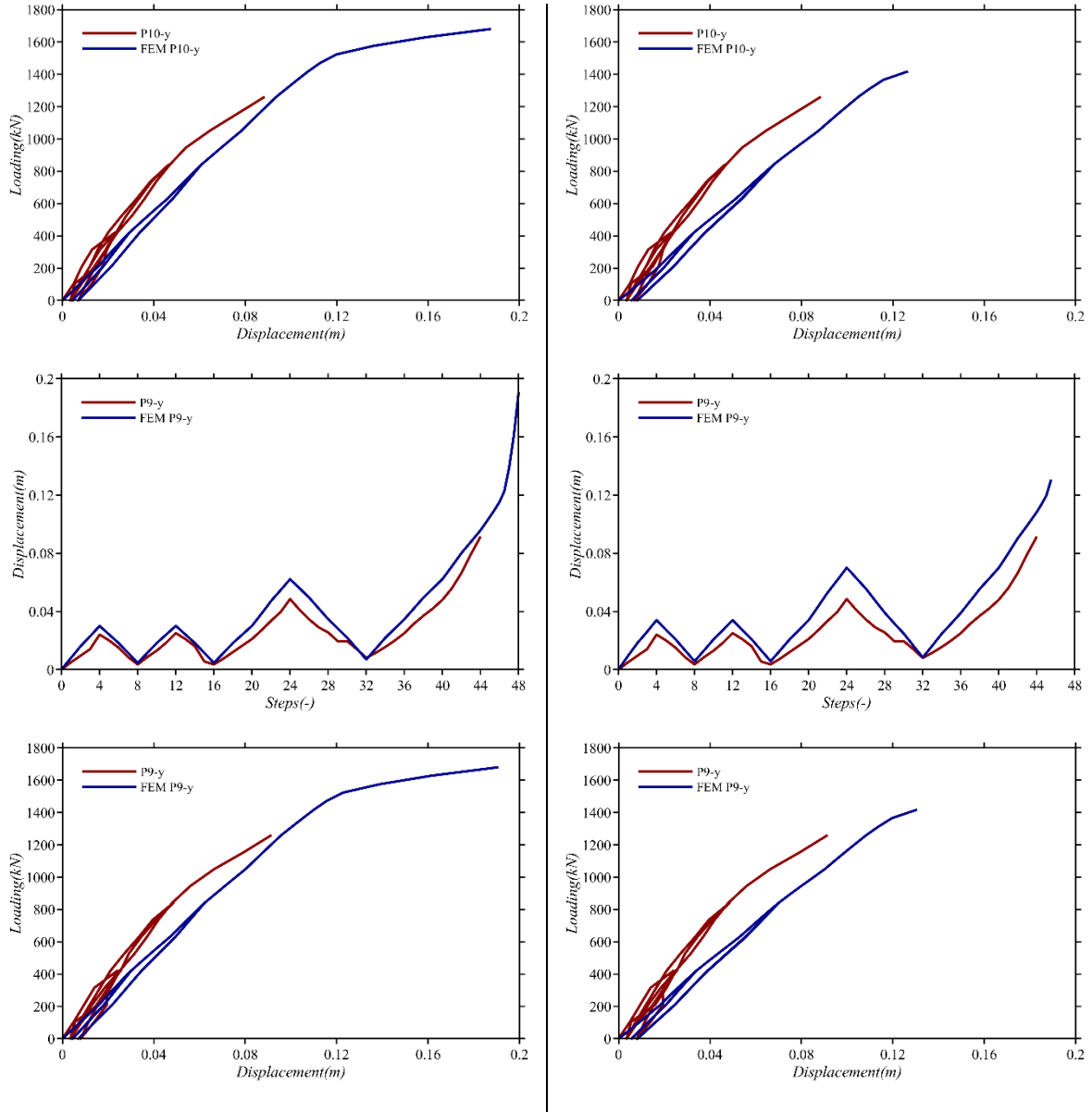


Figure 5.5. vertical displacement at P9 and P10, against the loading step and against the total load, experiment against a) Model with steel and b) Model with interface.

Figure 5.6 compares the transverse displacement of P9 and P10 with the experimental record. This component originates from the rotation of the section and is therefore the more sensitive to the modelling of the bearing. The interface model reproduces the measured transverse displacement closely, both in shape and in magnitude, whereas the steel model proves excessively stiff in this direction: when plotted against the total load, the transverse slope of the steel model exceeds that of the real beam, while the transverse slope of the interface model practically coincides with the measurement.

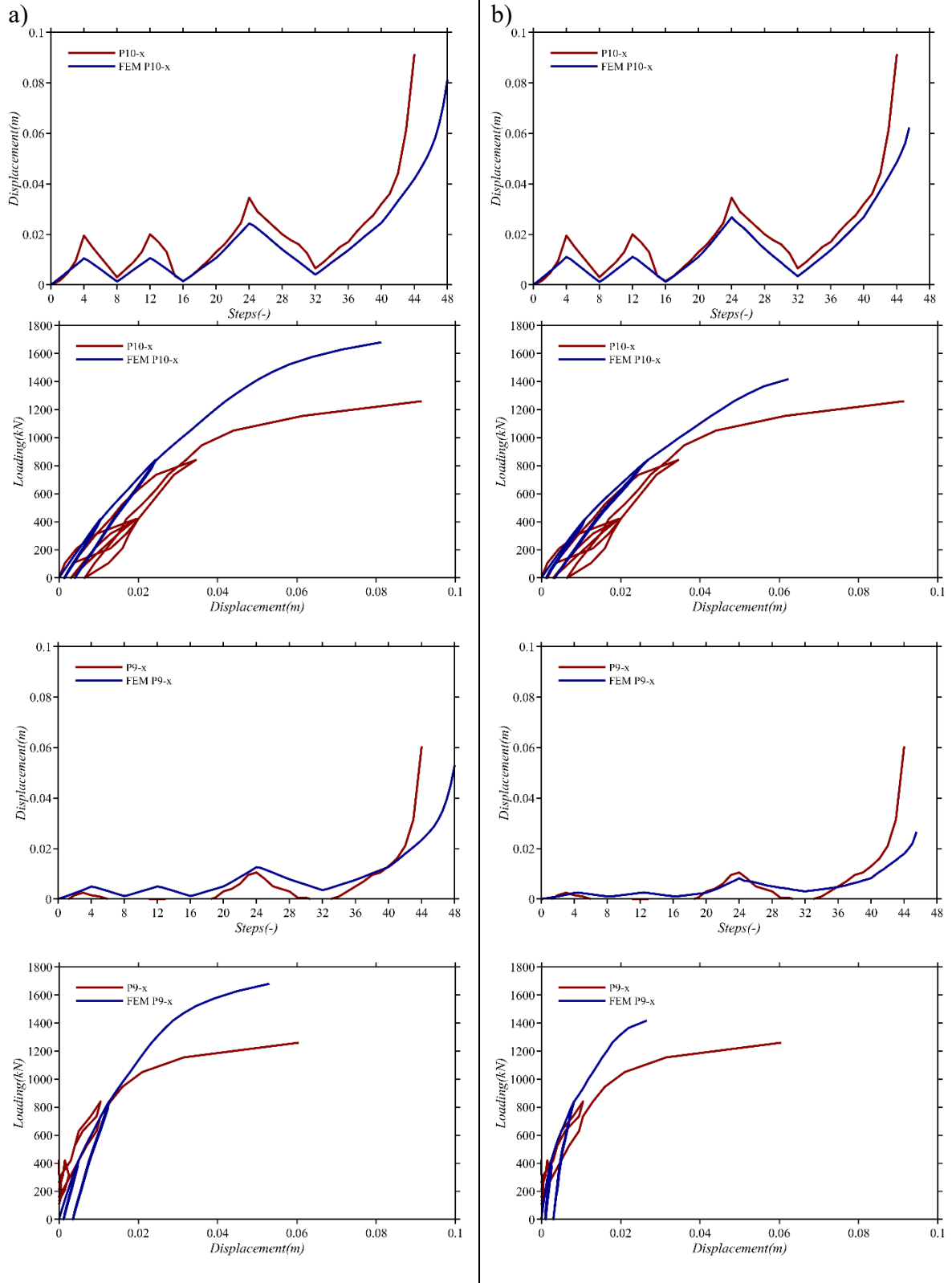
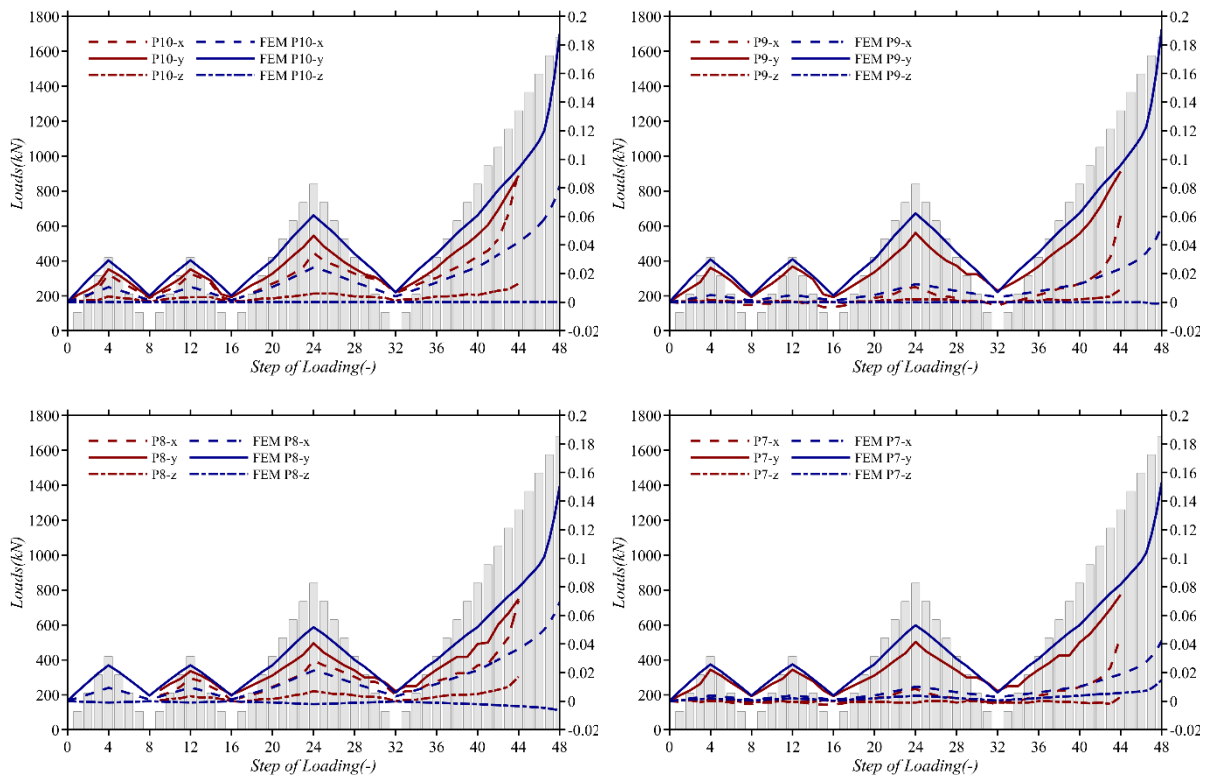


Figure 5.6. transverse displacement at P9 and P10, against the loading step and against the total load, experiment against a) Model with steel and b) Model with interface.

Taken together, these two components explain why both models are retained. The steel model reproduces the vertical displacement most faithfully, while the interface model reproduces the transverse displacement most faithfully, and since neither model is superior on both components, both are carried forward through the remainder of the comparison. Both models also share a slightly low vertical stiffness. This follows from the calibration of the bearing, since a stiffer bearing would improve the vertical match but would at the same time make the transverse response too stiff, so a moderate bearing stiffness was retained in order to balance the two directions.

Beyond P9 and P10, the comparison is extended to the remaining targets distributed along the beam. Figure 5.7 and Figure 5.8 collects the three displacement components, namely the vertical, the transverse and the longitudinal, at targets P1, P2, P6, P7, P8, P9, P10 and at their counterparts P1', P2', P6', P7' and P8' on the second alignment, each compared with its own experimental record. The targets located away from mid-span undergo smaller displacements, and at these positions the two models remain close both to the measurement and to each other, since they lie far from the bearing region in which the models differ.



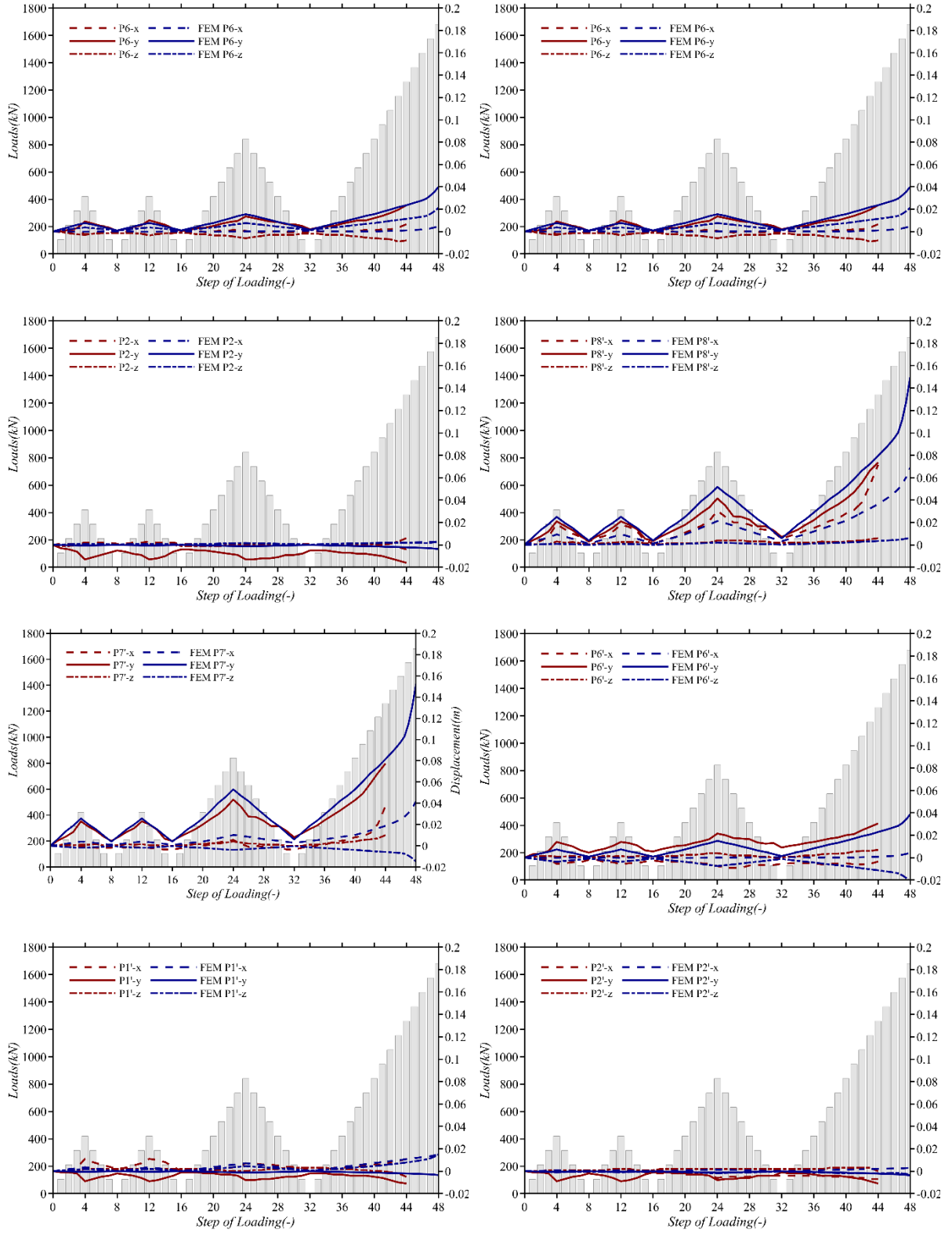
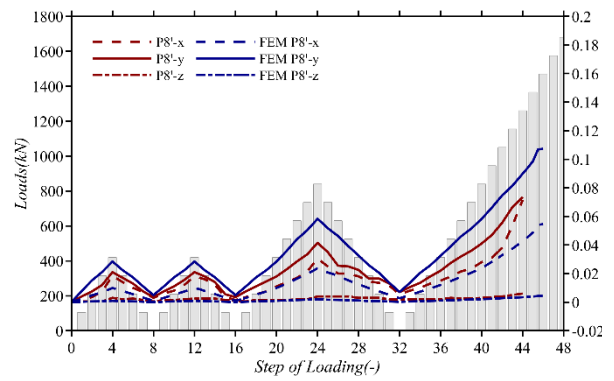
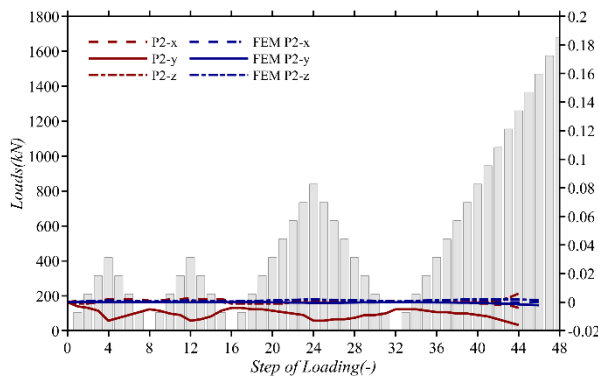
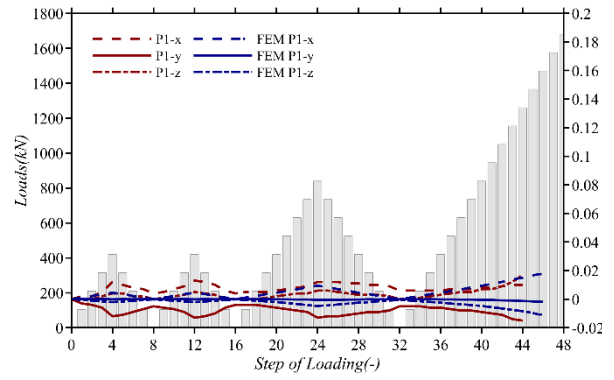
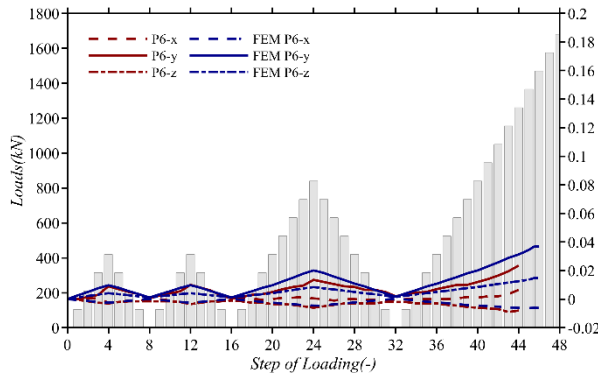
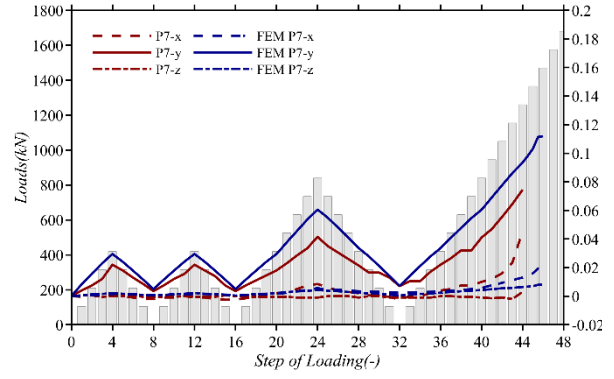
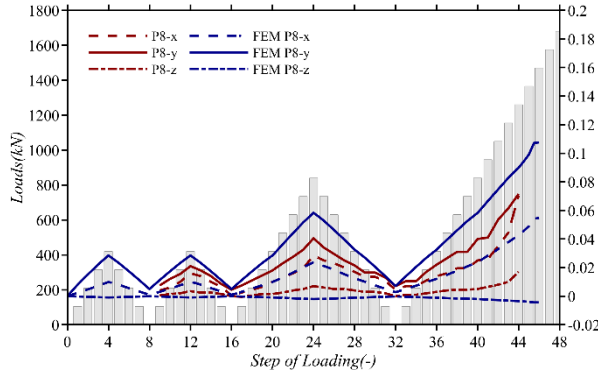
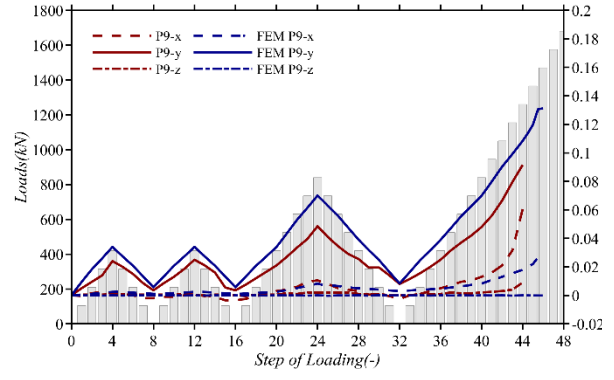
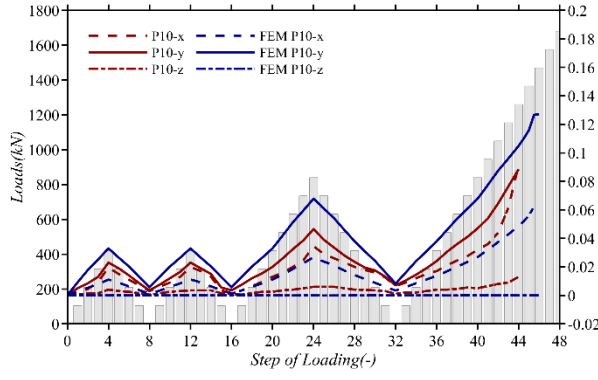


Figure 5.7. vertical, transverse and longitudinal displacement at targets P1, P2, P6, P7, P8, P9, P10 and P1', P2', P6', P7', P8', experiment against Model with steel.



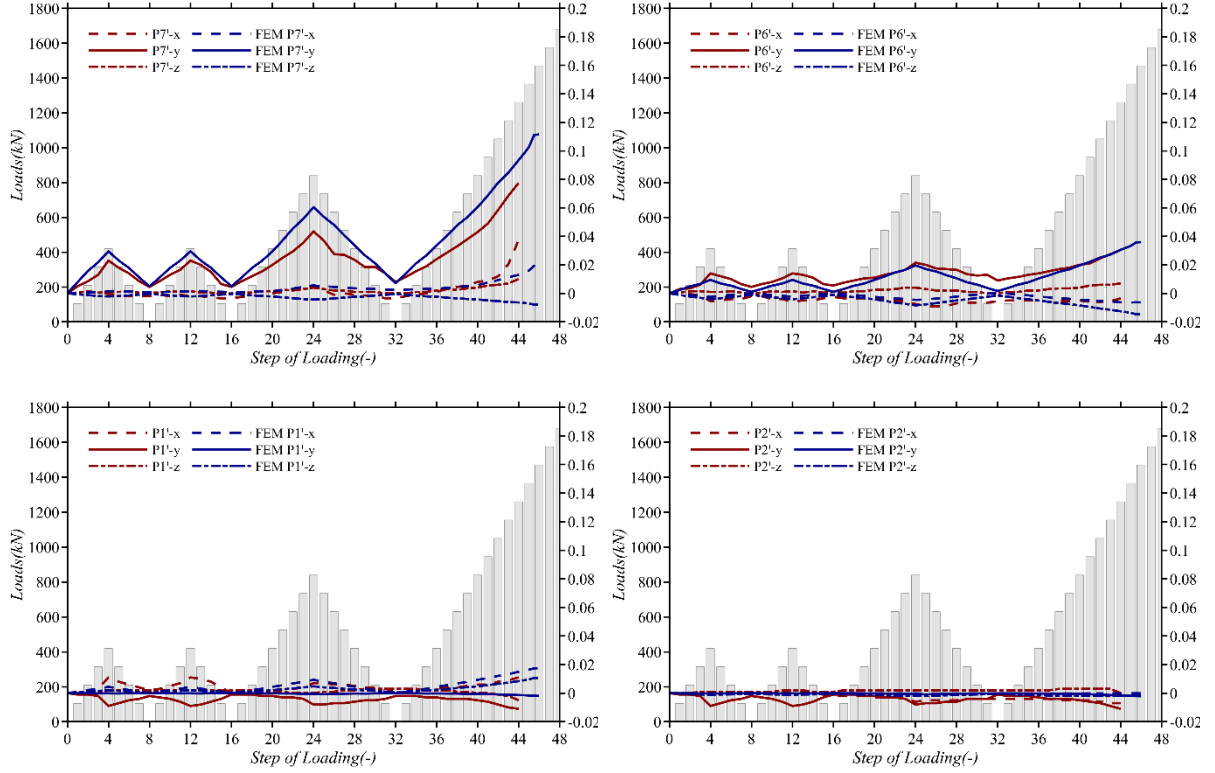


Figure 5.8. vertical, transverse and longitudinal displacement at targets P1, P2, P6, P7, P8, P9, P10 and P1', P2', P6', P7', P8', experiment against Model with interface.

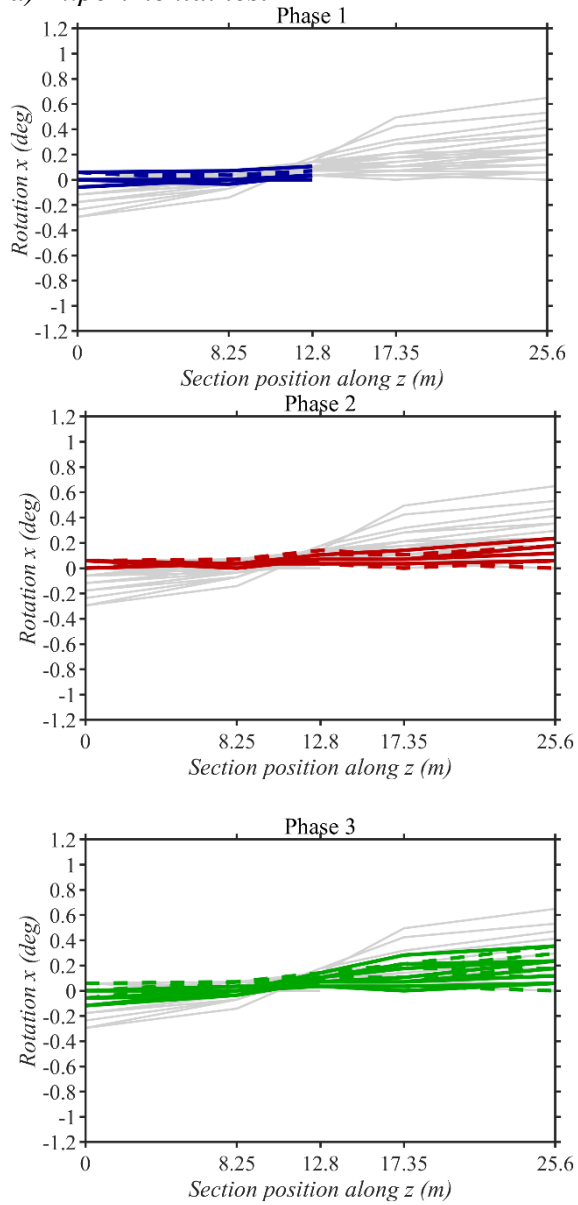
5.3 Rotation comparison

The rotation of the cross-section is the kinematic quantity most directly governed by the bearing, and it therefore provides a sensitive measure of the quality of the support model. It is evaluated at five sections distributed along the span. At each section the rotation is obtained as the difference between the displacement of the upper monitoring target and that of the lower target on the same section, divided by the vertical distance separating the two points. This difference is formed separately from the transverse and from the longitudinal components of the displacement, yielding two distinct rotations, one about the longitudinal axis and one about the transverse axis, which are reported and compared with the experiment [6] independently of one another.

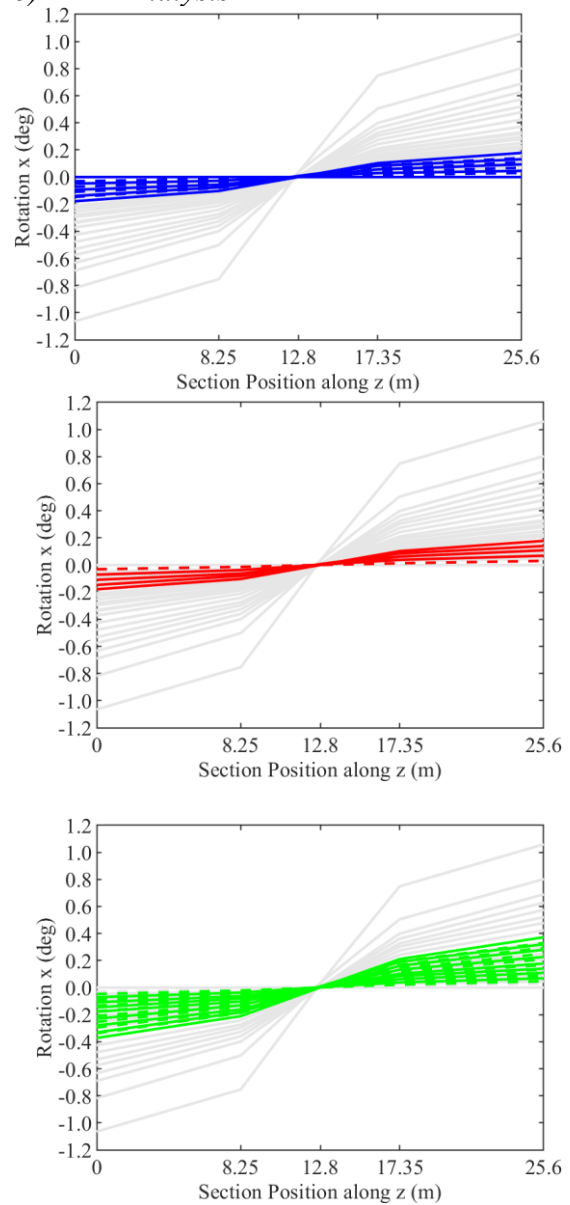
Figure 5.9 and Figure 5.10 compare the rotation about the transverse axis, that is, the flexural rotation of the section. This rotation is antisymmetric along the span and passes through zero at mid-span, since the slope of the deflected beam reverses sign at that location, and it likewise

grows through the four loading phases. Both models reproduce the antisymmetric distribution and the magnitude of this rotation with good accuracy, in agreement with the close match obtained for the vertical displacement in Section 5.2.

a) *Experimental test*



b) *FEM Analysis*



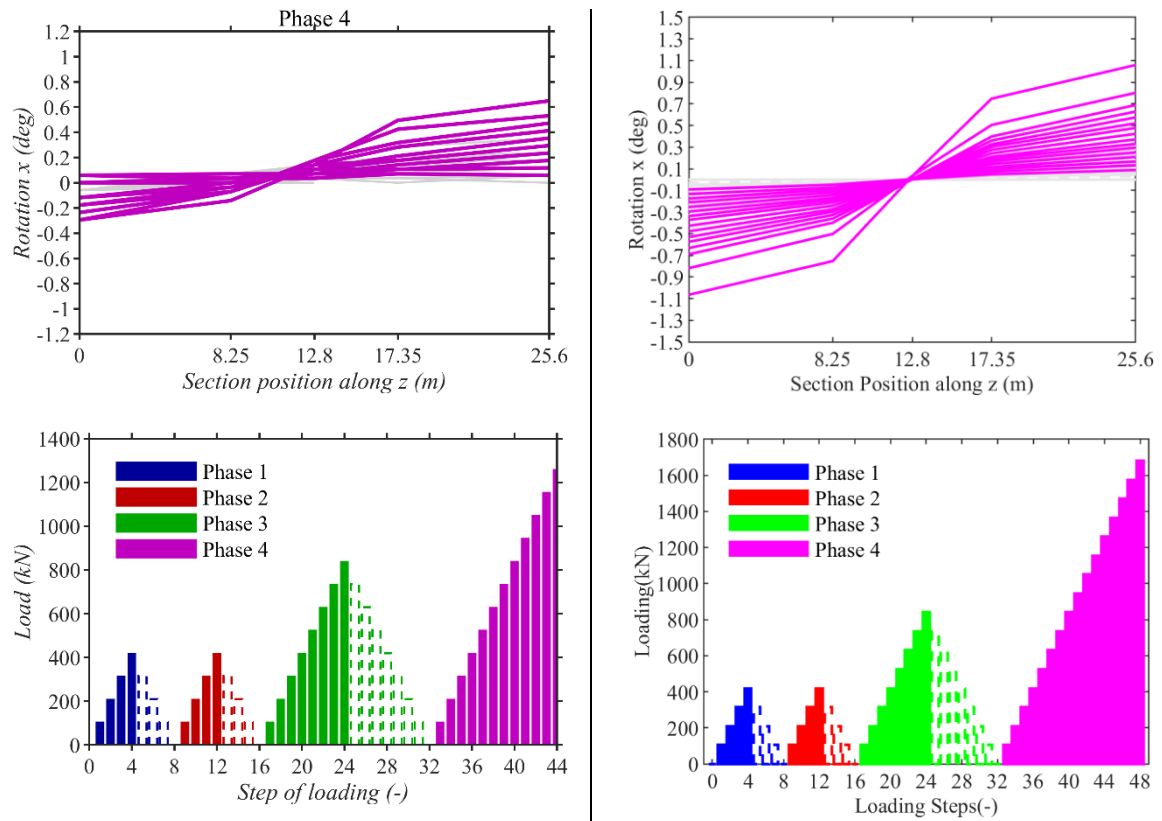
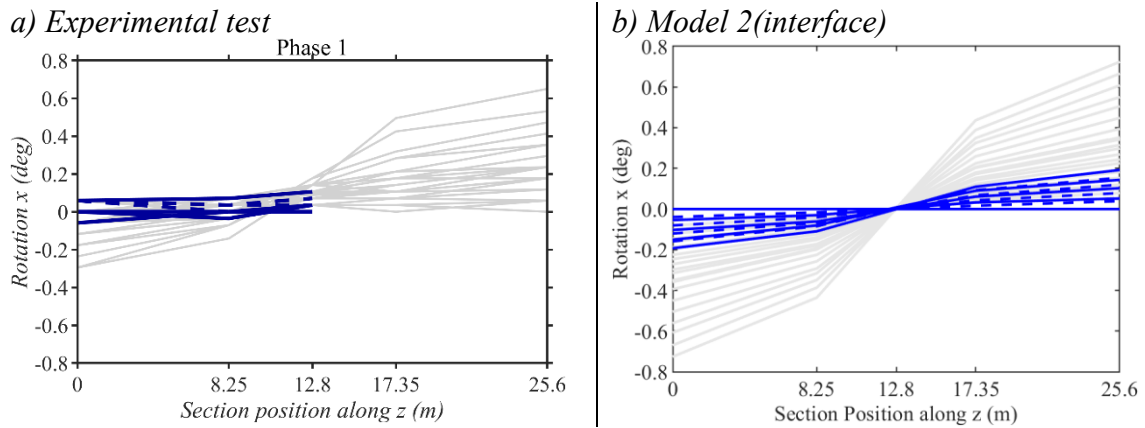


Figure 5.9. rotation about the transversal axis (flexural rotation) at five sections along the span, for the four loading phases, a) experiment against b) the FEM analysis of the model with steel as bearing material.



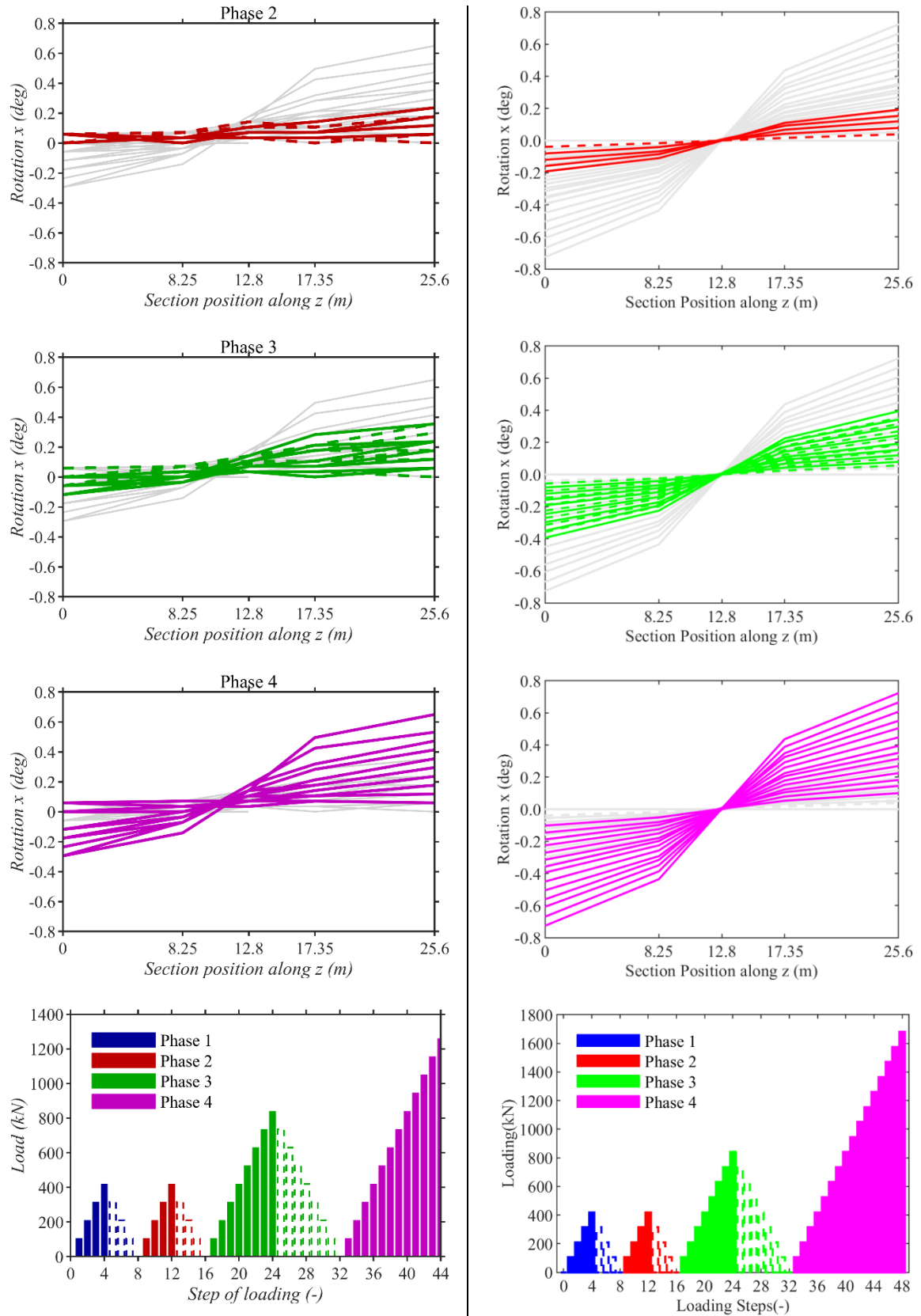
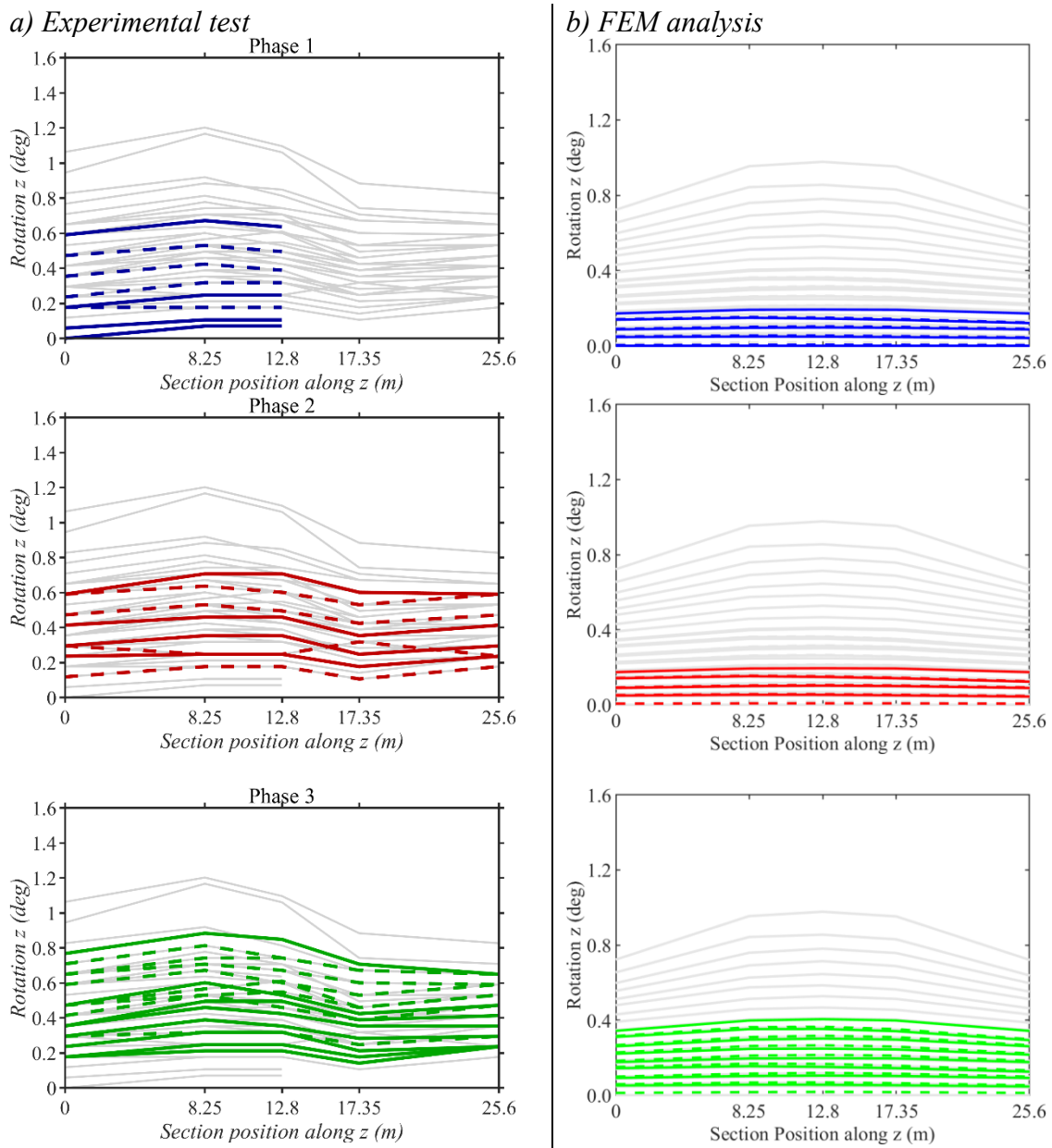


Figure 5.10. rotation about the transversal axis (flexural rotation) at five sections along the span, for the four loading phases, a) experiment against b) the FEM analysis of the model with Interface.

Figure 5.11 and Figure 5.12 compare the rotation about the longitudinal axis, that is, the torsional rotation of the section. This rotation retains a single sign along the beam, increases towards mid-span and grows progressively through the four loading phases. Both models reproduce this general shape, with the largest rotation occurring near mid-span. The numerical rotation is distributed more uniformly along the span, whereas the experimental rotation is more sharply concentrated in the central region, also the model with interface is, however, larger than the measured one.



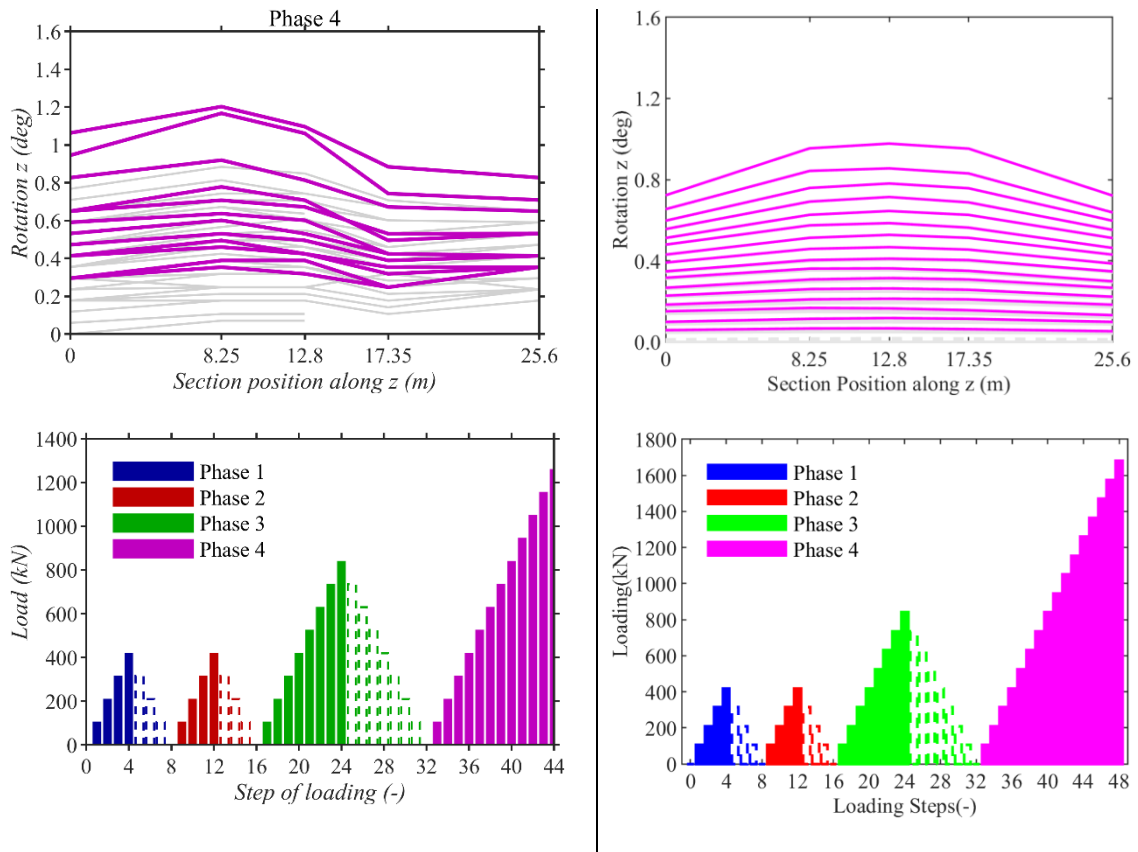
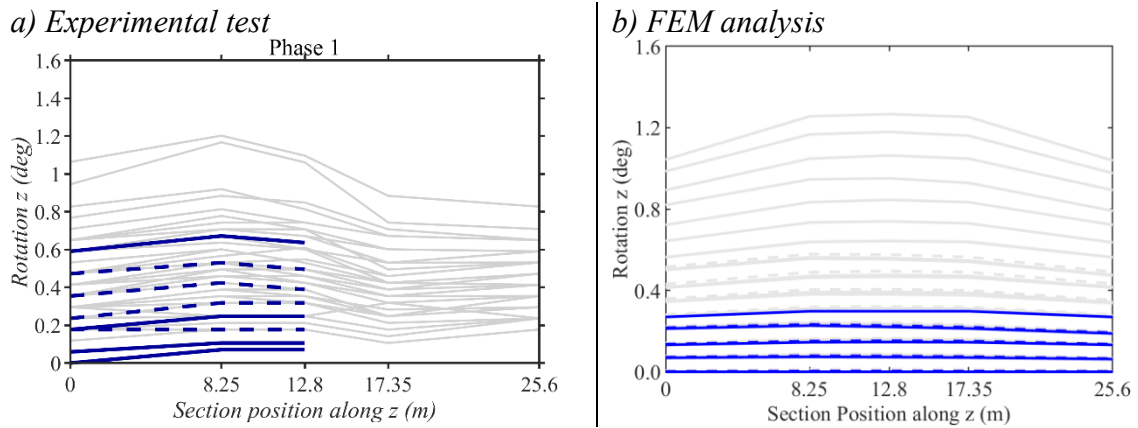


Figure 5.11. rotation about the longitudinal axis (torsion) at five sections along the span, for the four loading phases, a) experiment against b) the model with steel as bearing material.



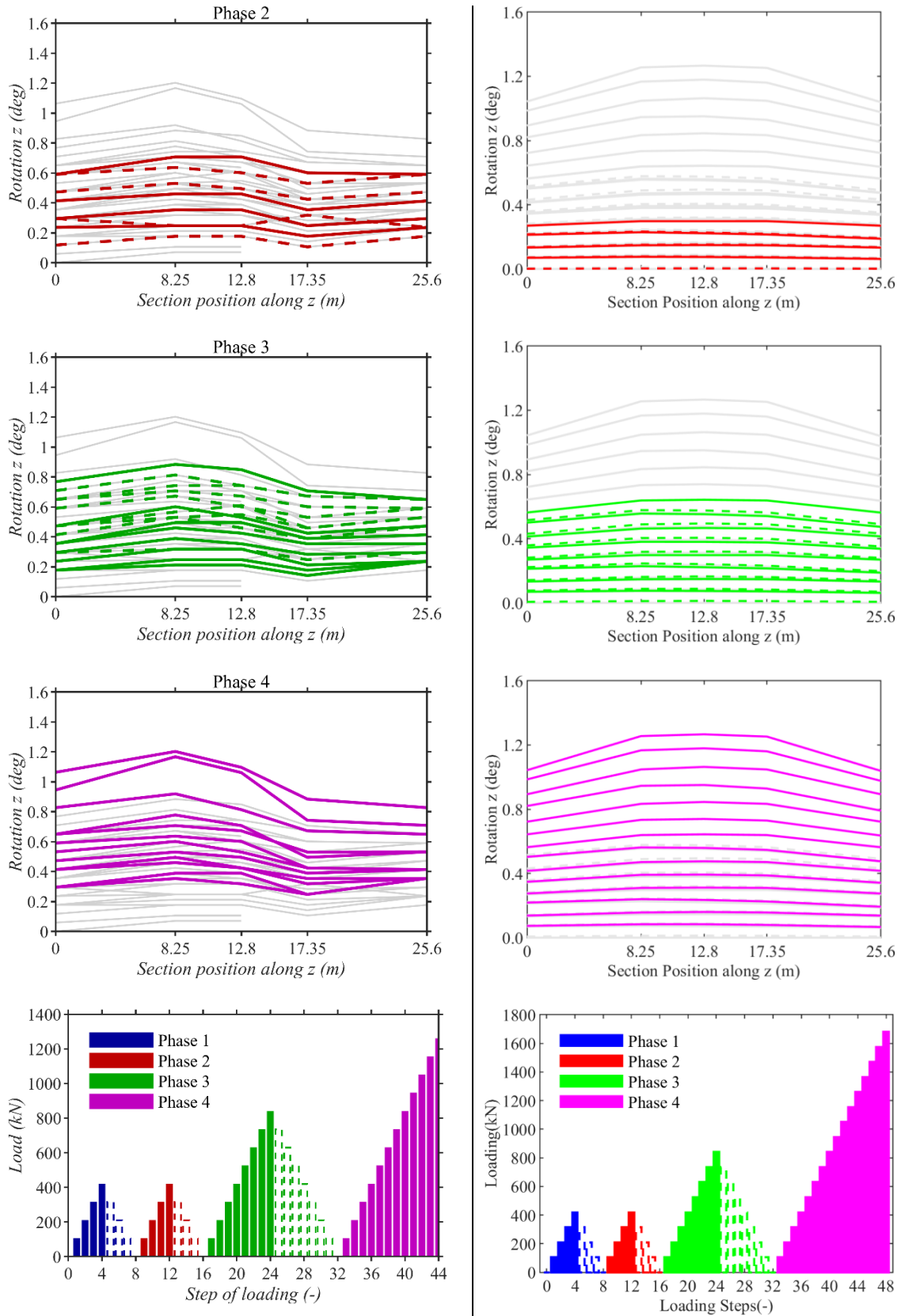


Figure 5.12. rotation about the longitudinal axis (torsion) at five sections along the span, for the four loading phases, a) experiment against b) the model with interface.

Since the rotation is derived from point displacements rather than measured directly, it inherits the scatter of those displacements, and small discrepancies in the transverse movements are amplified when the twist is computed.

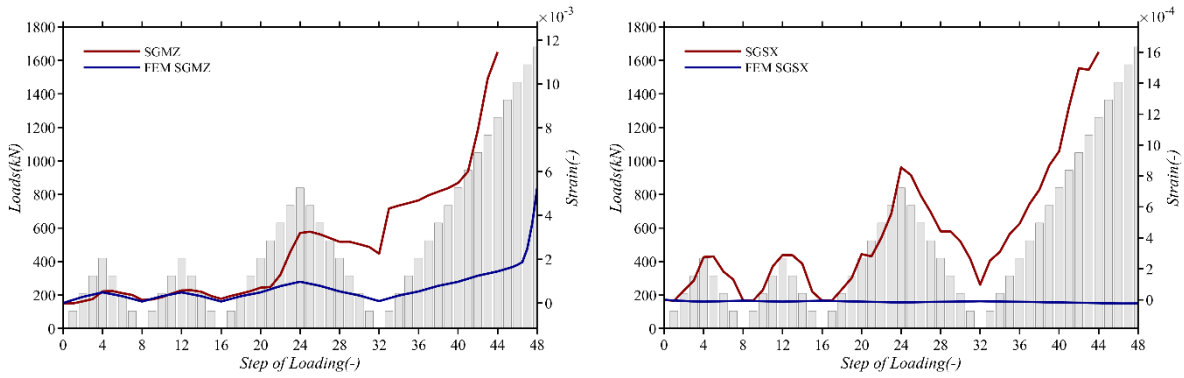
5.4 Reinforcement strain at the gauges

Two induction strain gauges were bonded to the bottom $\Phi 28$ reinforcement during the test, SGSX at one third of the span and SGMZ at mid-span [6]. They recorded the strain of the principal tensile reinforcement, that is, the bar layer that governs the flexural failure. In the model the monitors SGSX and SGMZ record the axial strain of the same bars at the same sections, as defined in Section 3.12, so that the numerical and measured strains refer to the same fibre.

Figure 5.13 compares the two gauges, numerical against measured, for both models. The yield strain of the bars is approximately 0.002, obtained as the ratio of $f_y = 397.2$ MPa to $E_s = 200$ GPa [6]. At mid-span the measured strain at SGMZ exceeds this value and continues to grow well beyond it as the load approaches collapse, confirming that the bottom bars yielded. The numerical strain reaches the yield value at the same section but remains below the measured peak, and the steel model approaches the measurement more closely than the interface model.

At the third point of the span the measured strain at SGSX increases with the load, whereas the corresponding numerical strain remains low and almost constant. The explanation is local in character. At that position the concrete cover over the bars had deteriorated in the years preceding the test, so that the real reinforcement carried a higher strain than would be expected in an intact section. The model represents a sound section at this location and does not reproduce the local damage, and it therefore underestimates the strain at SGSX.

a)



b)

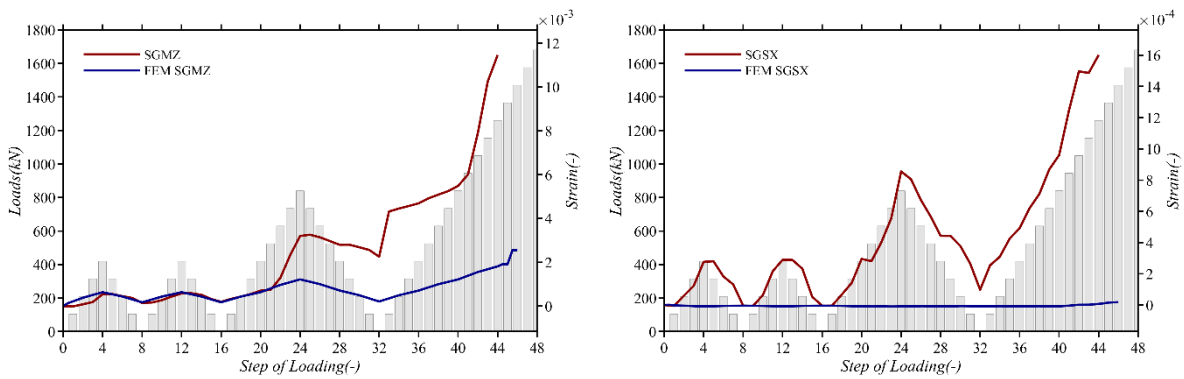


Figure 5.13.reinforcement strain at SGSX (third point) and SGMZ (mid-span) against the loading step, experiment against a) Model with Steel material and b) Model using interface.

The residual difference at SGMZ is consistent with the nature of the smeared crack model. A physical gauge is mounted on a single bar and records the high local strain at the particular crack that crosses it, whereas the model returns the strain averaged over the finite element. The comparison therefore confirms the flexural mechanism, and in particular the yielding of the bottom bars at mid-span, rather than the precise value of the peak strain. Since the gauges are located far from the bearing, the two models produce similar reinforcement strains, and the difference between them is small in comparison with their common difference from the measurement.

5.5 Conclusions

This thesis has developed a three-dimensional nonlinear finite element model of the 25.90 m reinforced concrete edge beam removed from a decommissioned viaduct on the A6 motorway, and has compared it with the load-to-collapse test performed on the same beam in 2021 [6]. The study falls within the scope of a Level 4 in-depth assessment under the 2020 Guidelines for existing bridges [3] in which a refined numerical model is validated against experimental evidence before being used to assess the safety of the structural element.

The response of the beam was found to be governed by its supports. The asymmetry of the edge section converted the vertical load into a combination of bending and torsion, and the measured displacements were controlled by the movements occurring at the Gerber saddles. Since a model based on ideal constraints cannot reproduce such measurements, the support was treated by means of a parametric study, which led to two final models. In the first, the bearing is retained as a steel volume with an equivalent modulus of 5 MPa in absence of any additional transverse restraint; in the second, the bearing pad is replaced by a Mohr-Coulomb interface with an equivalent modulus of 3.5 MPa, complemented by a linear transverse spring of 12 MPa kept active for the entire analysis.

Both models reproduce the measured behavior throughout the loading cycles and up to collapse. They yield the same deformed shape and the same crack pattern, with shear cracks near the supports and flexural cracks at mid-span, and they locate failure in the same region, that is, the differentiated web at mid-span. Both attain their limit at a load close to the experimental value of 1260 kN, while the reinforcement strain at mid-span exceeds the yield strain of the bottom bars, confirming the flexural nature of the failure. The agreement is, however, shared between the two models: the steel model reproduces the vertical displacement, whereas the interface model reproduces the transverse displacement and the rotation. Both models exhibit a slightly low vertical stiffness, which follows from the way the bearing was calibrated rather than from the beam itself. Increasing the stiffness of the bearing raises the apparent stiffness of the beam in both the vertical and the transverse direction, so that a stiffer bearing brings the vertical response closer to the measurement but makes the transverse response too stiff and moves it away from the measurement. The modulus of the bearing and the lateral restraint were therefore set to a moderate value, accepting a slightly low vertical stiffness in order to keep both the vertical and the transverse displacements reasonably close to the experiment, rather than improving one component at the expense of the other.

These results convey a clear engineering implication for the assessment. The flexibility of the support and the constructional details of the test influence the predicted displacements and rotations to an extent comparable with that of the section properties, so that a credible assessment of this type of beam cannot be based on idealized constraints. The measured collapse load was lower than the resistance predicted from the historical design data [6] and part of this difference is attributable to the presence of 19 bottom bars rather than the 20 indicated in the documentation. A validated model recovers both the actual capacity and the actual location of failure, which a hand calculation based on nominal data cannot provide, and this constitutes the practical value of the model for the real structure.

The model is nevertheless subject to a number of limitations identified during its development. A single constitutive law, derived from the $\Phi 28$ bars, was assigned to every bar diameter, since the tested bars represent the only experimental information available on the reinforcement. The rotating crack model adopts no shear retention, which is appropriate for the non-reversing load history of this test but would not be suitable for a fully reversed cyclic loading. The transverse spring of the interface model is linear and active throughout the analysis, a simplification imposed in part by the software, and it serves to control the free transverse slip of the interface; this spring was in fact necessary to maintain numerical stability, since its removal caused numerical overflow, so that the interface result is conditional on its presence. The isotropic bearing modulus couples the vertical and transverse stiffnesses of the pad, with the consequence that the two components could not be calibrated entirely independently, which is one of the reasons why no single model matches both. Finally, the bond between the bars and the concrete was assumed to be perfect, the local cover damage at the third point of the span was not modelled, and the smeared crack model returns averaged strains that underestimate the local strains recorded by the gauges.

These limitations also define the directions of future work. The first task is a mesh sensitivity study, since the discretization was fixed at a global size of 0.20 m and not varied, in order to confirm that the results are independent of the mesh in the sense of the benchmarking studies of fib bulletin 99 [42]. The introduction of a bond-slip law for the reinforcement, together with a local treatment of the damaged cover, would bring the computed gauge strains closer to the measured values. The validated model further constitutes a natural starting point for the application of a safety format, such as the estimation of a coefficient of variation of the resistance or of a global resistance factor [21], [22], so that a design value of the resistance, and not merely a mean response, can be derived for the assessment of the bridge. With the

bearing suitably calibrated, the model reproduces the measured behavior of the beam up to collapse and thereby provides a validated basis for the assessment of the decommissioned structure.

Bibliography

- [1] Italian Government, "Decreto-Legge 28 settembre 2018, n. 109 , Disposizioni urgenti per la città di Genova", Gazzetta Ufficiale della Repubblica Italiana, 2018.
- [2] G. Calvi et al., "Once upon a time in Italy: the tale of the Morandi Bridge", Structural Engineering International, vol. 29, no. 2, pp. 198-217, 2019.
- [3] Ministero delle Infrastrutture e dei Trasporti , Consiglio Superiore dei Lavori Pubblici, "Linee Guida per la classificazione e gestione del rischio, la valutazione della sicurezza ed il monitoraggio dei ponti esistenti", Decreto Ministeriale n. 578 del 17 dicembre 2020.
- [4] Ministero delle Infrastrutture e della Mobilità Sostenibili, "Aggiornamento delle Linee Guida per la classificazione e gestione del rischio, la valutazione della sicurezza ed il monitoraggio dei ponti esistenti", 2022.
- [5] J. R. Casas and J. D. Sobrino, "Geometric and material non-linear analysis of bridges", Engineering Structures, vol. 23, pp. 175-189, 2001.
- [6] Politecnico di Torino , DISEG, internal technical report on the load-to-collapse tests performed on the 25.90 m and 34.60 m bridge beams decommissioned from a viaduct on the A6 motorway, 2021.
- [7] Ministero dei Lavori Pubblici, "Normale n. 8 del 15 settembre 1933 , Carichi da considerarsi nel calcolo dei ponti per strade ordinarie", 1933.
- [8] Ministero dei Lavori Pubblici, "Circolare n. 6018 del 9 giugno 1945", 1945.
- [9] R. Cantieni, "Dynamic behaviour of highway bridges under the passage of heavy vehicles", EMPA Report No. 220, Swiss Federal Laboratories for Materials Testing and Research, 1992.
- [10] ANAS , Direzione Generale, "Circolare n. 820 del 15 marzo 1952", 1952.
- [11] Ministero dei Lavori Pubblici, "Circolare n. 384 del 14 febbraio 1962", 1962.
- [12] Ministero dei Lavori Pubblici, "Decreto Ministeriale del 2 agosto 1980 , Norme tecniche per la progettazione, esecuzione e collaudo dei ponti stradali", 1980.
- [13] Ministero dei Lavori Pubblici, "Decreto Ministeriale del 4 maggio 1990 , Aggiornamento delle norme tecniche per la progettazione, esecuzione e collaudo dei ponti stradali", 1990.
- [14] Ministero delle Infrastrutture e dei Trasporti, "Decreto Ministeriale del 14 gennaio 2008 , Norme Tecniche per le Costruzioni (NTC 2008)", 2008.

- [15] Ministero delle Infrastrutture e dei Trasporti, "Decreto Ministeriale del 17 gennaio 2018 , Aggiornamento delle Norme Tecniche per le Costruzioni (NTC 2018)", Gazzetta Ufficiale n. 42 del 20 febbraio 2018.
- [16] Ministero delle Infrastrutture e dei Trasporti, "Circolare n. 7 del 21 gennaio 2019 , Istruzioni per l'applicazione dell'Aggiornamento delle Norme Tecniche per le Costruzioni di cui al Decreto Ministeriale 17 gennaio 2018", 2019.
- [17] CEN, "EN 1991-2 , Eurocode 1: Actions on structures , Part 2: Traffic loads on bridges", European Committee for Standardization, Brussels, 2003.
- [18] fib , Federation Internationale du Beton, "fib Model Code for Concrete Structures 2010", Ernst & Sohn, Berlin, 2013.
- [19] D. L. Allaix, V. I. Carbone and G. Mancini, "Global safety format for non-linear analysis of reinforced concrete structures", Structural Concrete, vol. 14, no. 1, pp. 29-42, 2013.
- [20] V. Cervenka, "Reliability-based non-linear analysis according to fib Model Code 2010", Structural Concrete, vol. 14, no. 1, pp. 19-28, 2013.
- [21] V. Cervenka, "Global safety format for nonlinear calculation of reinforced concrete", Beton- und Stahlbetonbau, vol. 103, S1, pp. 37-42, 2008.
- [22] H. Schlune, K. Gylltoft and M. Plos, "Safety formats for non-linear analysis of concrete structures", Magazine of Concrete Research, vol. 64, no. 7, pp. 563-574, 2012.
- [23] M. Sykora, M. Holicky and J. Markova, "Verification of existing reinforced concrete bridges using the semi-probabilistic approach", Engineering Structures, vol. 56, pp. 1419-1426, 2013.
- [24] CEN, "EN 1990 , Eurocode 0: Basis of structural design", European Committee for Standardization, Brussels, 2002 (with amendments to 2010).
- [25] CEN, "EN 1992-1-1 , Eurocode 2: Design of concrete structures , Part 1-1: General rules and rules for buildings", European Committee for Standardization, Brussels, 2004.
- [26] CEN, "EN 1992-2 , Eurocode 2: Design of concrete structures , Part 2: Concrete bridges , Design and detailing rules", European Committee for Standardization, Brussels, 2005.
- [27] CEN/TC 250, "prEN 1990:2023 , Basis of structural and geotechnical design", draft of the second-generation Eurocode 0, including prAnnex E on assessment of existing structures, 2023.

- [28] M. Sykora, M. Holicky, M. Prieto and P. Tanner, "Uncertainties in resistance models for sound and corrosion-damaged RC structures according to EN 1992-1-1", *Materials and Structures*, vol. 48, pp. 3415-3430, 2015.
- [29] fib , Federation Internationale du Beton, "fib Model Code for Concrete Structures 2020", Ernst & Sohn, Berlin, 2024.
- [30] JCSS , Joint Committee on Structural Safety, "Probabilistic Model Code", 2001 (updated periodically, www.jcss-lc.org).
- [31] ISO, "ISO 2394 , General principles on reliability for structures", International Organization for Standardization, 2015.
- [32] ISO, "ISO 13822 , Bases for design of structures , Assessment of existing structures", International Organization for Standardization, 2010.
- [33] M. Holicky and M. Sykora, "Reliability of concrete bridges considering model and statistical uncertainties", *Structural Concrete*, vol. 12, no. 3, pp. 162-170, 2011.
- [34] P. Castaldo, D. Gino and G. Mancini, "Safety formats for non-linear finite element analysis of reinforced concrete structures: discussion, comparison and proposals", *Engineering Structures*, vol. 193, pp. 136-153, 2019.
- [35] ASME, "Guide for Verification and Validation in Computational Solid Mechanics", ASME V&V 10-2006, American Society of Mechanical Engineers, 2006.
- [36] P. J. Roache, "Verification and Validation in Computational Science and Engineering", Hermosa Publishers, Albuquerque, 1998.
- [37] W. L. Oberkampf and C. J. Roy, "Verification and Validation in Scientific Computing", Cambridge University Press, 2010.
- [38] D. L. Allaix and V. I. Carbone, "Numerical discretization error analysis for the global safety format of NLFEA", *Structural Concrete*, vol. 14, no. 1, pp. 43-52, 2013.
- [39] CEN, "EN 13791 , Assessment of in-situ compressive strength in structures and precast concrete components", European Committee for Standardization, Brussels, 2019.
- [40] fib bulletin 45, "Practitioners' guide to finite element modelling of reinforced concrete structures", Federation Internationale du Beton, Lausanne, 2008.
- [41] V. Cervenka, L. Jendele and J. Cervenka, "ATENA Program Documentation , Part 1: Theory", Cervenka Consulting, Prague, 2025.
- [42] fib bulletin 99, "Benchmarking of nonlinear analysis methods for concrete structures", Federation Internationale du Beton, Lausanne, 2021.

- [43] V. Cervenka, J. Cervenka and R. Pukl, "ATENA , a tool for engineering analysis of fracture in concrete", *Sadhana*, vol. 27, no. 4, pp. 485-492, 2002.
- [44] Cervenka Consulting, "ATENA Program Documentation , User's Manual for ATENA Engineering (ATENA Pre)", Prague, 2025.
- [45] Cervenka Consulting, "ATENA Program Documentation , User's Manual for ATENA Studio", Prague, 2025.
- [46] H. Kupfer, H. K. Hilsdorf and H. Rusch, "Behavior of concrete under biaxial stresses", *ACI Journal Proceedings*, vol. 66, no. 8, pp. 656-666, 1969.
- [47] P. Menetrey and K. J. Willam, "Triaxial failure criterion for concrete and its generalization", *ACI Structural Journal*, vol. 92, no. 3, pp. 311-318, 1995.
- [48] Z. P. Bazant and B. H. Oh, "Crack band theory for fracture of concrete", *Materials and Structures*, vol. 16, no. 93, pp. 155-177, 1983.
- [49] H. Schlune, M. Plos and K. Gylltoft, "Improved bridge evaluation through finite element model updating using static and dynamic measurements", *Engineering Structures*, vol. 31, no. 7, pp. 1477-1485, 2009.

Acknowledgements

This thesis is the end of a long journey, and I did not walk it alone. I want to thank the people who supported me along the way.

To my mother and father, thank you for everything. You gave me the courage to leave home, to immigrate and to explore the world, and you stood behind every choice I made, even when it took me far from you. I carry your love with me every day.

To my fiancé, Tohid, thank you for standing beside me at every step. Your kindness and your constant support have made my life happier and easier, and you believed in me when I doubted myself. This achievement is as much yours as it is mine.

To my sisters, thank you for being the people I turn to in any difficulty. Knowing that you are always there, whatever happens, has given me a strength that is hard to put into words.

A special thank you goes to Giovanni. His help was invaluable, and his calm, steady presence made a real difference. He shared his knowledge generously and was always ready to answer my questions, even the smallest ones.

I am also deeply grateful to my supervisor, Professor Diego Gino, for his guidance throughout this work. He was always kind and patient, and his calm advice helped me move forward whenever I felt stuck. Finally, I thank my co-supervisors, Professor Paolo Castaldo and Elena, for their help, their careful feedback and the time they gave to this project.