

FEM simulation of static loading test of the Omega beam

Petr Bílý¹, Alena Kohoutková¹ and Petr Jedlinský²

¹Faculty of Civil Engineering, Czech Technical University in Prague, Thákurova 7,
166 29 Praha 6, Czech Republic

²EUROVIA Services, s.r.o, Národní 138/10, 110 00 Praha 1, Czech Republic

E-mail: petr.bily@fsv.cvut.cz

Abstract. The paper deals with a FEM simulation of static loading test of the Omega beam. Omega beam is a precast prestressed high-performance concrete element with the shape of Greek letter omega. Omega beam was designed as a self-supporting permanent formwork member for construction of girder bridges. FEM program ATENA Science was exploited for simulation of load-bearing test of the beam. The numerical model was calibrated using the data from both static loading test and tests of material properties. Comparison of load-displacement diagrams obtained from the experiment and the model was conducted. Development of cracks and crack patterns were compared. Very good agreement of experimental data and the FEM model was reached. The calibrated model can be used for design of optimized Omega beams in the future without the need of expensive loading tests. The calibrated material model can be also exploited in other types of FEM analyses of bridges constructed with the use of Omega beams, such as limit state analysis, optimization of shear connectors, prediction of long-term deflections or prediction of crack development.

1. Introduction

When an unusual new type of a load-bearing element for bridges is designed with the use of a non-traditional material, it is sometimes the case that a static loading test of 1:1 scale model is performed before practical application to verify the expected behavior of the element. Such tests are usually quite expensive and therefore it is suitable to use their results for calibration of numerical models that will allow avoiding the large-scale tests in the future. A properly calibrated numerical model can fully replace the static loading test.

In this paper, the process of numerical modeling of the static loading test of so called Omega beam (for more information, see paper “Lightweight HPC beam OMEGA“ by Sýkora et al. published in the same proceedings as this paper [7]) is described and the results of the model and the test are compared. Omega beam is a precast prestressed high-performance concrete (HPC) element with the shape of Greek letter omega. Omega beam was designed as a self-supporting permanent formwork member for construction of girder bridges. It exploits high durability of HPC [6]. FEM software ATENA Science version 5.3.4 [1] was used for numerical modeling.

2. Static loading test

A prototype of Omega beam (see Figures 1 and 2) was produced and subjected to static loading test in the age of 17 days. The test took place in the precast concrete production plant. The load was applied to the beam through the steel crossbars loaded by hydraulic presses propped against a group of



concrete panels (see Figure 3). The total weight of the panels was 30 tons, the panels rested on concrete supports.



Figure 1. View of the end part of the Omega beam.

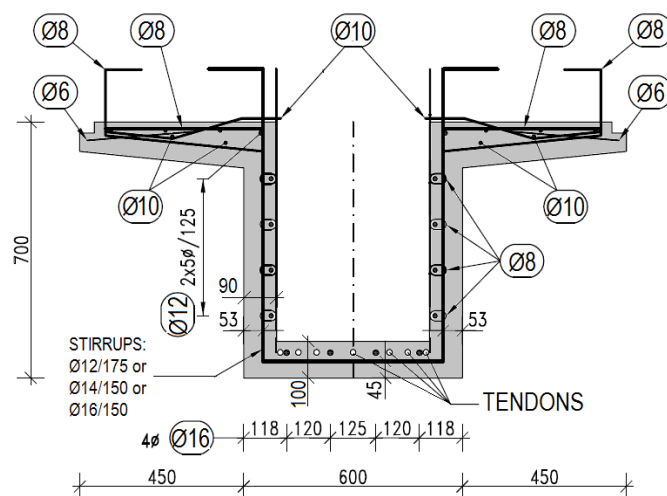


Figure 2. Section of the Omega beam with description of the reinforcement.

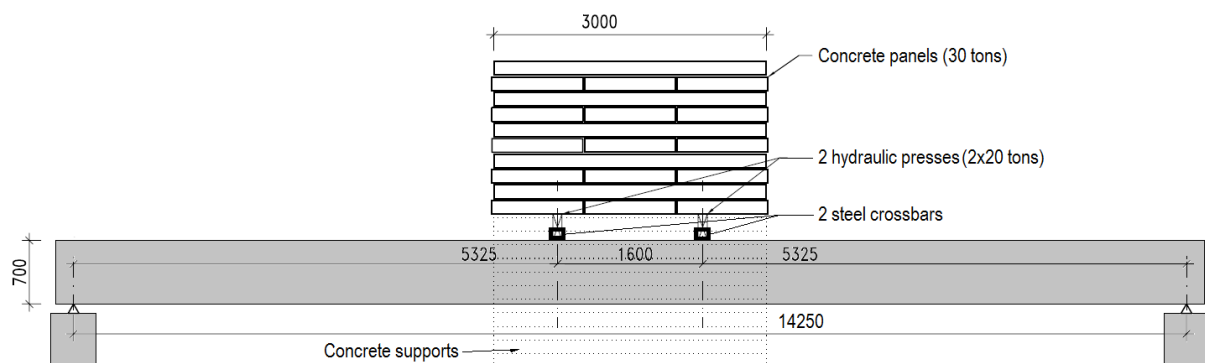


Figure 3. Scheme of the static loading test of the Omega beam.

The schedule of the static loading test is given in Table 1. The first load step simulated the loads from in-situ concreting of the monolithic part of the bridge structure. The second one represented the decompression of the lower part of the cross-section. The third one corresponded to the anticipated moment of reaching the tensile strength of concrete, i.e. the moment when the first cracks occur and the non-linear behavior of the structure begins. It was planned to interrupt the test before reaching the theoretical ultimate load-bearing capacity as the prototype was meant for exploitation in a real structure. The loads in particular steps were calculated by hand calculations performed according to ČSN EN 1992-1-1 [2].

Table 1. Load steps of the static loading test.

Load step no.	Load step	Load [kN]	Load [%]
1	In-situ concreting	2×85	60
2	Decompression	2×89	63
3	Tensile strength	2×111	78
4	End of the test	2×142	100
-	Theoretical capacity	2×172	121

3. FEM model

The simulation of the loading test was performed in ATENA Science version 5.3.4 [1]. Creep module was used to include the creep and shrinkage of concrete between the production of the prototype and the static loading test. Newton-Raphson calculation method was applied. The model was prepared in GiD 11 preprocessing tool.

3.1. Geometry

To optimize the computation process, symmetrical quarter of the beam was modelled (Figure 4). The model was supported by a hinge in the middle of the supporting plate (orange colour in Figure 4), horizontal constraints were applied in the direction perpendicular to the section planes (green and yellow colour in figure 4) to simulate the influence of cut parts of the structure. The load was introduced through a part of the crossbeam (red arrow in Figure 4). Regular hexahedral mesh with five elements in thickness of the Omega beam was used, the length of the elements was 50 mm in the central part of the beam and 115 mm in the rest. Each bar of the longitudinal reinforcement was divided in 100 linear finite elements, each stirrup was divided in 10 linear finite elements.

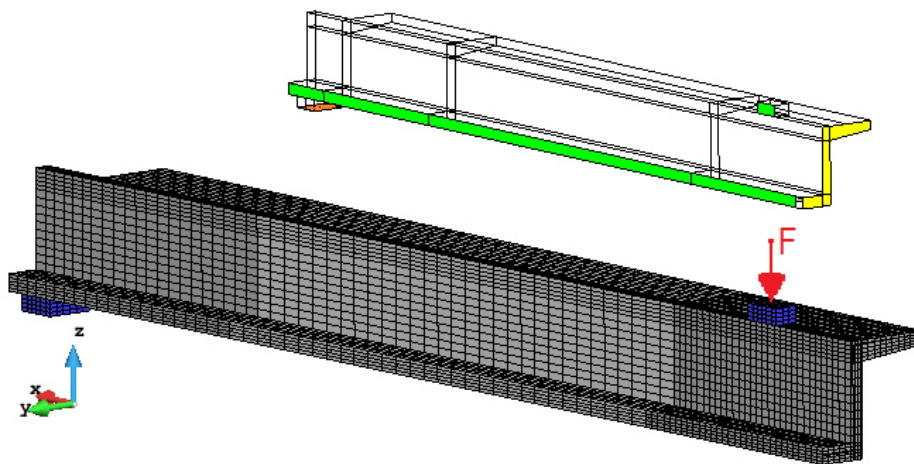


Figure 4. Model of the Omega beam. Lower part: FEM mesh and the load. Upper part: boundary conditions (explanation in the paragraph above).

3.2. Materials

Mean cylindrical compressive strength of concrete $f_{cm} = 68.2$ MPa was obtained from the results of laboratory tests. Other properties were calculated according to the formulae given in the *fib* Model Code 2010 [3], see Table 2. Concrete class C60/75 XF2, XD1 was used.

Material model CC3DNonLinCementitious2 was utilized for the basic deformation behavior of the material. The model combines constitutive models for tensile (fracturing) and compressive (plastic) behavior. The fracture model is based on the classical orthotropic smeared crack formulation and crack band model. It employs Rankine failure criterion. The hardening/softening plasticity model is based on Menétrey-Willam failure surface [4].

To model the time-dependent behavior of concrete, material model CCmodelB3Improved based on B3 model created by Bažant and Baweja [5] was exploited. The parameters used in the model are summarized in Table 3. This model considers autogenous and drying shrinkage, basic and drying creep, temperature effect on creep and coupling between creep and fracture. B3 model was also used to calculate the time development of material properties of concrete from the properties in the age of 28 days.

The structure was reinforced by B500 B bars and St 15.7-1660/1860 tendons with very low relaxation. Mean material parameters were used in the model (see Tables 4 and 5). CCReinforcement

material model with bilinear stress-strain diagram was exploited for embedded reinforcement bar elements.

Table 2. Mechanical properties of concrete

Property	Label	Value	Unit	Formula [3]
Mean cylindrical compressive strength	f_{cm}	68.2	MPa	-
Mean elastic modulus	E_{cm}	40.76	GPa	$21.5(f_{cm}/10)^{1/3}$
Mean axial tensile strength	f_{ctm}	4.4	MPa	$2.12\ln(1+f_{cm}/10)$
Relative fracture energy	G_F	156	Nm ⁻¹	$73f_{cm}^{0.18}$
Tension stiffening factor	F_t	0.4	-	-
Bulk density	ρ_c	2285	kg.m ⁻³	-

Table 3. Material parameters for B3 model.

Property	Value	Unit
Cement type	CEM I 52.5 R	-
Cement content	457.6	kg.m ⁻³
Water to cement ratio	0.41	-
Aggregate to cement ratio	3.70	-
Effective thickness of the beam	0.094	m
Relative ambient humidity	70	%
Cross-section shape factor	1.0	-
Curing conditions	air	-
Curing time	2	days

Table 4. Mechanical properties of reinforcement bars.

Property	Label	Value	Unit
Mean yield limit	f_{ym}	550	MPa
Elastic modulus	E_s	200	GPa
Ultimate strength	f_{um}	594	MPa
Ultimate elongation	ε_{um}	5	%
Bulk density	ρ_s	7850	kg.m ⁻³
Bar diameters	$\emptyset$	6 – 16	mm

Table 5. Mechanical properties of tendons.

Property	Label	Value	Unit
Mean yield limit	f_{pym}	1826	MPa
Elastic modulus	E_p	193	GPa
Ultimate strength	f_{pum}	1972	MPa
Ultimate elongation	ε_{pum}	3.5	%
Bulk density	ρ_s	7850	kg.m ⁻³
Cross-sectional area of 1 tendon	A_{pl}	150.5	mm ²

3.3. Loads

The model was loaded in several steps according to Table 6. The forces in steps 3 – 6 represented the static loading test described in section 2. As one quarter of the beam was modelled, the applied loads are equal to one quarter of the total load in the test. Some values of the forces are slightly different from those in Table 1 as the real process of the test did not agree exactly with the plan (the cracks occurred later). The forces in steps 3 – 8 were applied in several increments in order to reach the convergence of the solution process. The model was loaded up to the ultimate limit state (ULS).

The self-weight was applied to the model by means of the defined bulk densities of the materials (concrete, reinforcement).

The prestress was introduced as the initial strain of tendons $\varepsilon_{p,0}$. The value was calculated using the Hooke's law and considering the final stress of tendons after straining $\sigma_{p,0} = 1462$ MPa, anchorage slip $p = 2.4$ mm and total length of the tendons 37 557 mm:

$$\varepsilon_{p,0} = -\frac{\sigma_{p,0} - \frac{p}{L} E_p}{E_p} = -\frac{1462 - \frac{2,4}{37557} \cdot 193000}{193000} = -7.511 \cdot 10^{-3} \quad (1)$$

The loss due to elastic shortening of concrete after application of the prestress was modeled by the software. The same applies to the progress of creep and shrinkage of the structure, including the respective losses in prestress. The loss due to relaxation of tendons was calculated by hand according to [2] and entered to the model as a reduction of the initial strain of tendons $\Delta\varepsilon_{p,r} = 1.38 \times 10^{-4}$.

Table 6. Load steps

Step no.	Load step	Time [days]	New applied loads	Number of increments [-]	Total force in the end of the step [kN]
0	Production, curing	0.0 – 2.1	-	-	-
1	Prestressing	2.1 – 2.2	Self-weight, prestress	5	-
2	Aging, losses in prestress	2.2 – 17.0	Creep, shrinkage, losses in ps.	5	-
3	In-situ concreting	17.0 – 17.01	0.5×85 kN	5	0.5×85
4	Decompression	17.01 – 17.02	0.5×4 kN	5	0.5×89
5	Tensile strength	17.02 – 17.03	0.5×36 kN	10	0.5×125
6	End of the test	17.03 – 17.04	0.5×24 kN	5	0.5×149
7	Theoretical capacity	17.04 – 17.05	0.5×23 kN	5	0.5×172
8	ULS	17.05 – 17.065	0.5×75 kN	15	0.5×247

4. Results and discussion

The load-displacement curves obtained from the static loading test and the numerical model are presented in Figure 5 and Table 7. Very good overall agreement between the test and the basic material model with the parameters defined before (denoted as ATENA) was reached.

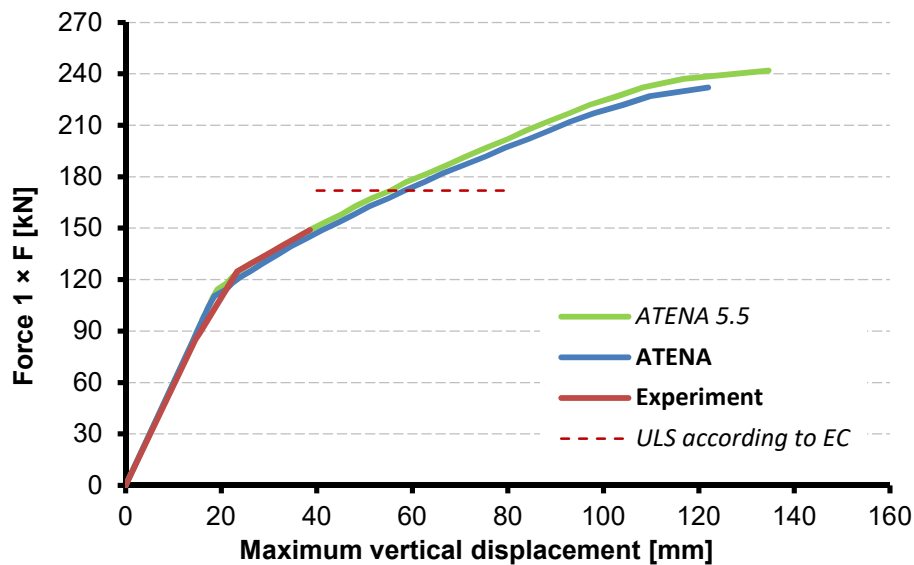


Figure 5. Comparison of load-displacement curves.

Table 7. Comparison of displacements from the experiment and the models.

Step no.	Load step	Total force in the end of the step[kN] $0.5 \times F$	Maximum vertical displacement [mm]		
			Experiment	ATENA	ATENA 5.5
3	In-situ concreting	0.5×85	14.6	14.2	14.2
4	Decompression	0.5×89	15.5	14.8	14.8
5	Tensile strength	0.5×125	23.4	26.1	23.7
6	End of the test	0.5×149	38.6	41.2	38.6
7	Theoretical capacity	0.5×172	-	58.4	55.4
8	ULS	0.5×232	-	121.9	108.3
9	ULS ver. 5.5	0.5×242	-	-	134.6

Certain disagreement is evident at the beginning of the non-linear behavior of the model. The first cracks occurred in the model as early as at the load of $F = 111$ kN, while no cracks were observed during the test until $F = 125$ kN. The width of the first cracks in the model was less than 0.02 mm. Cracks that thin would have no practical meaning in the real structure, but they influenced the stiffness of the very sensitive numerical model.

An attempt was made to further improve the affinity of the curves in the non-linear area by varying material parameters (tensile strength, fracture energy, tensile stiffening factor). The best result was reached when the tensile strength of concrete was increased to 5.5 MPa (the curve denoted as ATENA 5.5 in Figure 5). The tensile strength of the material was not measured directly by tests. This means that the real tensile strength could have been higher than the theoretical one considered in the basic model. The value of 5.5 MPa is less than 95 % quantile of tensile strength of C60/75 concrete ($f_{ctk,0.95} = 5.7$ MPa according to [2]), therefore it is realistic. However, there is no experimental evidence to justify such a high tensile strength and the value is not very likely according to the experience of the authors. Therefore, the ATENA 5.5 model is not recommended for further simulations.

The development of cracks is documented in Figures 6 – 11.

- The first cracks were observed during the test at the load of $F = 125$ kN when four cracks of approximately 0.1 mm appeared in the central area of the beam (Figure 6). At the same load, six cracks of 0.06 – 0.08 mm occurred in the corresponding area of the model (Figure 7).

- At the load of $F = 141$ kN, seven cracks of the width between 0.1 and 0.2 mm were registered during the test. No new cracks exceeding the width of 0.1 mm appeared in the model, the width of the six cracks in Figure 8 was between 0.1 and 0.12 mm.
- At the end of the test, the beam was disturbed by eight cracks 0.2 – 0.25 mm wide, while there were eight cracks approximately 0.1 – 0.15 mm wide in the model (Figure 9).
- Crack pattern at the theoretical ULS is in the Figure 10, the widest cracks had 0.2 mm.
- The extensive disruption of the beam at the ULS of the model is represented by Figure 11, with the width of the cracks reaching 0.5 mm.

As a result, it can be said that the crack pattern of the model corresponds to the reality, but the model tends to underestimate the width of the cracks and overestimate the number of the cracks at the same time. The calculated crack widths are dependent on the finite element mesh size. Theoretically, better agreement between the cracks in the model and the test could be reached by increasing the size of the elements in the x direction, but this could lead to problems with convergence of the solution process. Therefore the amendment was not applied.

The ultimate limit state of the ATENA model was reached at the load of $F = 232$ kN. At this moment, the highest compressive stress in concrete in the middle of the beam was $\sigma_{cc} = 45$ MPa, the tendons were slightly behind the yield limit ($\sigma_p = 1850$ MPa) and the reinforcement bars were exactly on the yield limit ($\sigma_s = 552$ MPa).

Further small increase of the applied load accompanied by plastic deformation of the reinforcement and large vertical displacements of the model would be possible, up to the moment of reaching the ultimate strength of one of the materials. The Newton-Raphson method used for the calculation did not allow simulating this situation; the Arc-Length method is not available in the Creep module of ATENA Science software. In any case, finding the load and displacement at the final ULS would have little practical meaning.



Figure 6. First cracks in the test (4×0.1 mm, $F = 125$ kN).

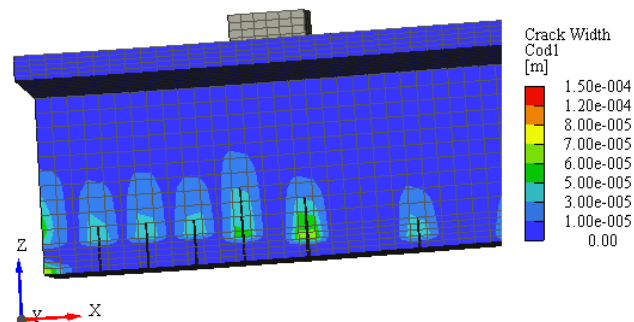


Figure 7. Cracks in the model at $F = 125$ kN. Width is between 0.06 – 0.08 mm.

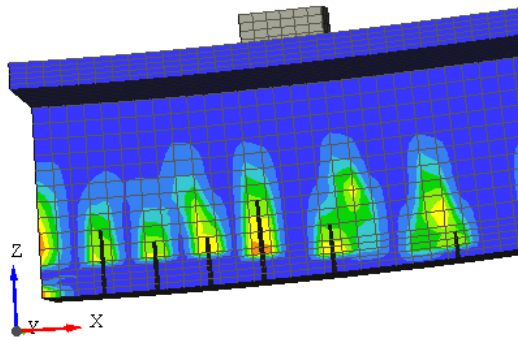


Figure 8. Cracks in the model at $F = 141$ kN. Width is between 0.1 – 0.12 mm. The same legend as in Figure 7 applies.

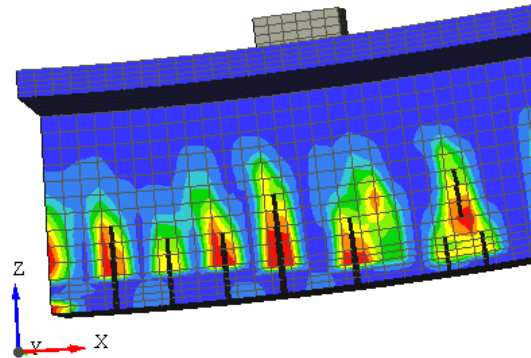


Figure 9. Cracks in the model at $F = 149$ kN. Width is between 0.1 – 0.15 mm. The same legend as in Figure 7 applies.

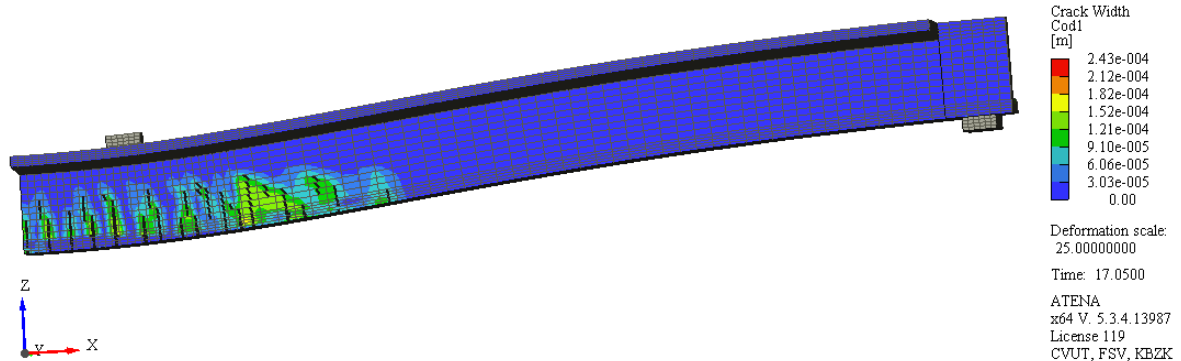


Figure 10. Deflection and cracks in the model at $F = 172$ kN (theoretical ULS). Width of the cracks is between 0.1 – 0.2 mm.

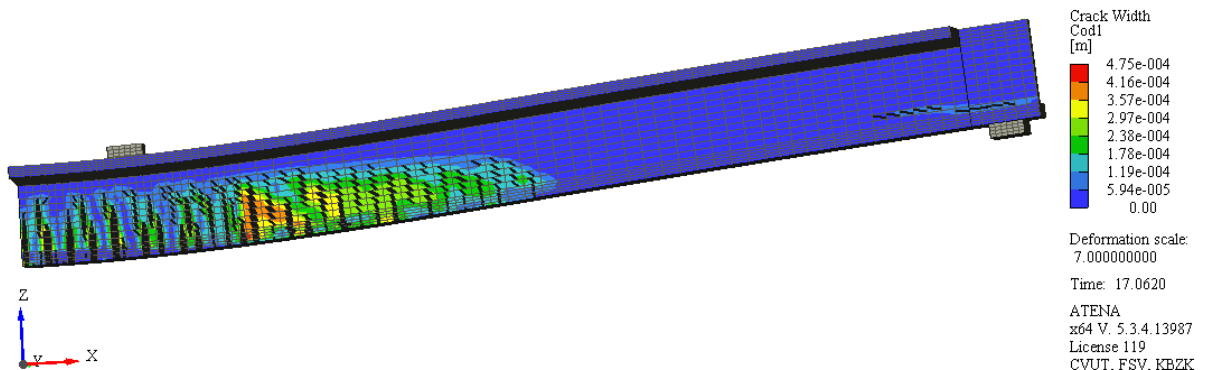


Figure 11. Deflection and cracks in the model at $F = 232$ kN (ULS of the model). Width of the cracks is between 0.1 – 0.5 mm.

5. Conclusion

Very good overall agreement of load-displacement curves from the experiment and the FEM model was reached. Further improvement of the model would be probably possible if the real tensile strength of the material was known. The crack patterns obtained from the model corresponded to the reality, but the model tended to underestimate the width of the cracks and overestimate the number of the cracks at the same time.

The calibrated model can be used for design of optimized Omega beams in the future without the need of expensive loading tests. The calibrated material model can be also exploited in other types of FEM analyses of bridges constructed with the use of Omega beams, such as limit state analysis, optimization of shear connectors, prediction of long-term deflections or prediction of crack development.

Acknowledgements

This paper was prepared thanks to the support of the Technology Agency of the Czech Republic (TAČR), project “Centre for Effective and Sustainable Transport Infrastructure (CESTI)” (no. TE01020168), and the Science Foundation of the Czech Republic (GAČR), project “Analysis of the relations between the microstructure and macroscopic properties of ultra-high performance concretes” (no. 17-19463S).

References

- [1] Červenka Consulting 2016 ATENA x64 Version 5.3.4. Available online at <<http://www.cervenka.cz/products/atenal>>
- [2] Český normalizační institut 2006 Design of concrete structures - Part 1-1: General rules and rules for buildings *Eurocode 2, Part 1-1 (ČSN EN 1992-1-1)*
- [3] Walraaven J et al. 2013 *fib Model Code for Concrete Structures 2010* (Berlin: Ernst&Sohn)
- [4] Pisano A A 2012 An Algorithm Approach for Peak Load Evaluation of Structural Elements Obeying a Menetrey-William Type Yield Criterion *Electronic Journal of Differential Equations* **167** pp 1-9
- [5] Bažant Z P and Baweja S 2000 Creep and Shrinkage Prediction Model for Analysis and Design of Concrete Structures: Model B3 *Adam Neville Symposium: Creep and Shrinkage – Structural Design Effects* ed A Al-Manaseer (Farmington Hills, Michigan, USA: ACI)
- [6] Fládr J, Bílý P and Vodička J 2016 Experimental Testing of Resistance of Ultra-high Performance Concrete to Environmental Loads *Procedia Engineering* **151** pp 170-176
- [7] Sýkora M, Jedlinsky P and Komanec J 2017 Lightweight HPC beam OMEGA *IOP Conference Series: Materials Science and Engineering (MSE) – Proceedings of BESTInfra 2017 conference* (Prague, Czech Republic) to be published