Červenka Consulting s.r.o.
Na Hrebenkach 55
150 00 Prague
Czech Republic
Phone: +420 220 610 018
E-mail: cervenka@cervenka.cz
Web: http://www.cervenka.cz

ATENA 2023
Program Documentation
Part 3-4

Example & Validation Manual
ATENA with CeSTaR 2 Module

Project result TM0100059-V4

Written by

Jan Červenka, Tomáš Altman,

Zdeněk Janda, Pavel Pálek and **Radomír Pukl**

Prague, December 2022

Acknowledgements:

The software was developed with partial support of ***TAČR DELTA 2 Programme***

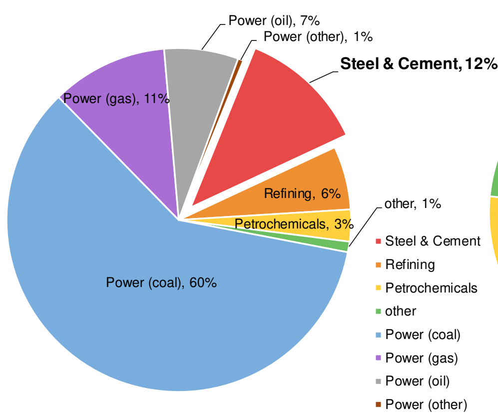
("DELTA 2 Funding programme for applied research, experimental development and innovation", project No. TM01000059 ***CeSTaR 2*** *– "Reducing material demands and enhancing structural capacity of multi-spiral reinforced concrete columns - advanced simulation and experimental validation")*

CONTENTS

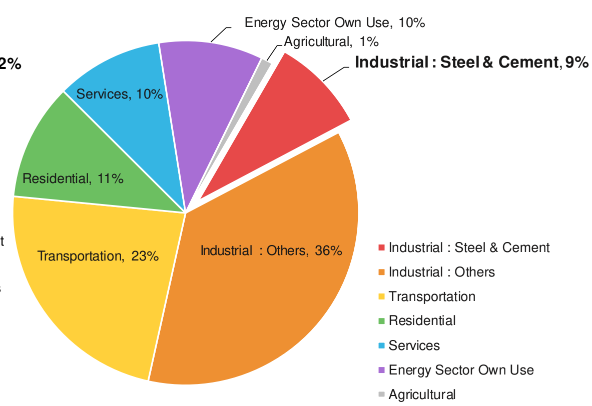
1. INTRODUCTION

This document contains the description of examples and validation problems that are included in the installation of the new module ATENA CeSTaR-2, which was developed during an international collaborative research project CeSTaR-2 „Reducing material demands and enhancing structural capacity of multi-spiral reinforced concrete columns - advanced simulation and experimental validation“. The project was supported by the DELTA-2 research program of the Czech Technological Agency.

The project motivation can be traced back to the report by Andrea Larson [1], carbon dioxide (CO_2) accounts for 77% of total green-house gas emission. And, the CO_2 emission from steel and cement industries occupies 12% of the total CO_2 emission (Fig. 1a). Moreover, according to the statistics of energy consumption in the first five months of 2019 published by Energy Bureau of Department of Economics of Taiwan [2], as shown in Fig. 1(b), it can be seen that the industrial sector consumes the largest amount of energy, accounting for 45% of the total energy consumption. The steel and cement industries accounts for 20% of the energy consumed by the industrial sector or for 9% of the total energy consumption. Cement production in the Czech Republic per capita is about 20% lower than in Taiwan. It is comparable to the quantity produced in Germany. Therefore, savings in the use of steel and concrete are crucial in reducing the CO_2 emission and energy consumption, promoting a greener environment for the place we live.



(a)

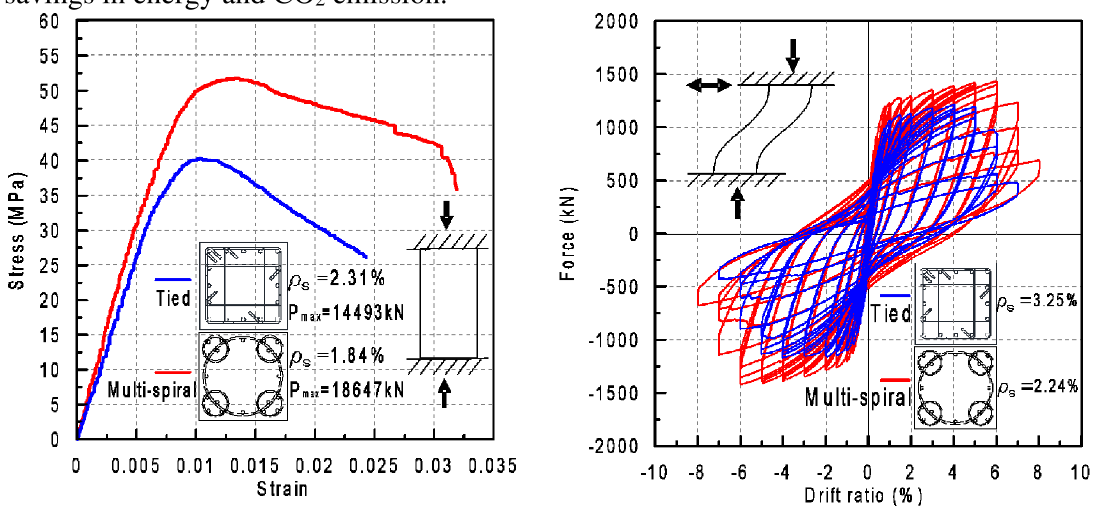


(b)

Figure 1: (a) Sources of CO_2 emissions [1]; and (b) Sources of energy consumption [2]

It has been shown by previous studies [3][4] that the use of multi-spiral reinforcement (MSR) in square or rectangular columns can significantly save the amount of steel for transverse reinforcement and yet can still achieve a higher structural performance than conventional tie reinforcement. A higher structural performance means a further save in steel reinforcement and concrete can be made for a given structural performance. Figure 2(a) shows test results of columns subjected to monotonically increasing axial compressive load [3]. The multi-spiral column has an amount of transverse reinforcement only 80% the amount used in a conventional tied column but still shows a

29% higher axial strength than the conventional tied column. Figure 2(b) shows the test results of columns subjected to lateral cyclic loading [4]. The multi-spiral column used only 69% the amount of transverse reinforcement used in the conventional tied column but still showed an 18% increase in lateral strength and a 59% increase in energy dissipation. These test results have demonstrated that concrete confined by multi-spiral reinforcement as a new form of confined concrete material can reduce the use of concrete and steel as compared with conventional confined concrete and hence promote savings in energy and CO_2 emission.



(a) (b)

Figure 2: Comparison between conventional tied column and multi-spiral column: (a) testing using monotonic axial load; (b) testing using cyclic loading test.

From past earthquakes in Taiwan, it has been shown that near-fault ground motions can cause severe damage to bridge columns due to the high velocity pulse and a large, one directional ground displacement as shown in Fig. 3. The damages as shown in Fig. 3 not only can be a destructive column failure but also can be a large permanent displacement of the column. Both can result in the demolition of the bridge, significantly threatens the lives of road users and causes negative impact on the society. The multi-spiral reinforcement will be utilized to enhance the structural performance of bridge columns to resist the destructive effects of near-fault ground motions. The superior confining effect of multi-spiral reinforcement is expected to provide enhanced protection to concrete to minimize the damage of concrete during strong ground motions. Moreover, high-strength strands functioning as elastic elements will be used to provide self-centering capability to minimize the permanent displacement of the column after the near-fault ground motion. This will ensure not only safety but also functionality of bridges right after the earthquake. This new concept is referred to as Advanced multi-spiral reinforcement (AMSR) layout.

These trends should be supported by cooperation between computer modeling and experimental research. Experimental research could serve as a basis for identification of appropriate input data of computer models and adjustment/development of the material models, and also for verifying the results provided by computer models of structures or structural members.

The software packages developed in the Czech Republic, namely ATENA software from Červenka Consulting and OOFEM by CTU will be extended for simulation of the investigated structural parts under complex loading conditions, which will increase their capabilities and open new potential markets and applications, in particular in locations and countries threatened by earthquakes.

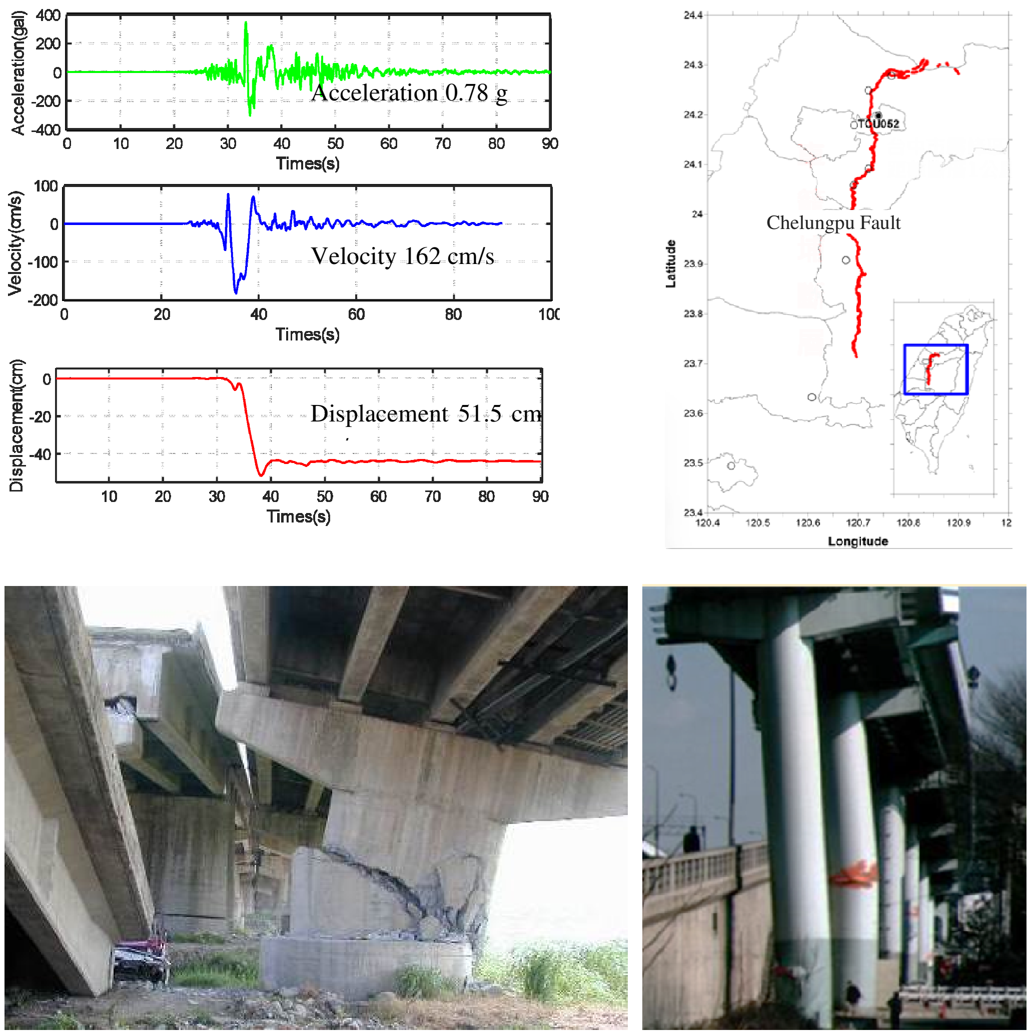


Figure 3: Severe damages to bridge columns caused by near-fault ground motions in Taiwan

2. STATIC ANALYSIS

This chapter contains 10 examples of static analysis using the program **ATENA 2023 CeSTaR2**.

2.1 Column CSCF

In this example the usage of the new CeSTaR 2 Module for preparation of the model of a concrete column with multispiral reinforcement for the analysis in the ATENA solution software is demonstrated.

File name: column_CSCF.pre

2.1.1 Geometry

Geometry is presented in the following figures (Figure 4 and Figure 5).

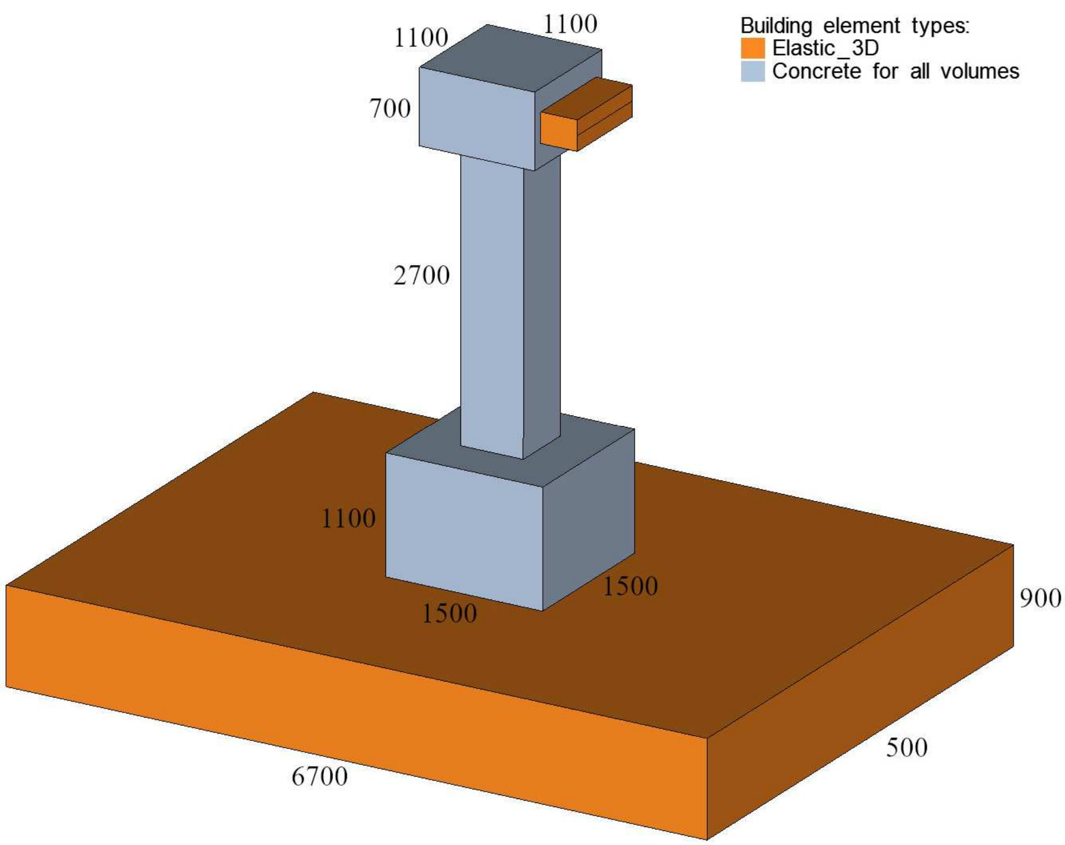


Figure 4: Solid model. Dimensions in mm

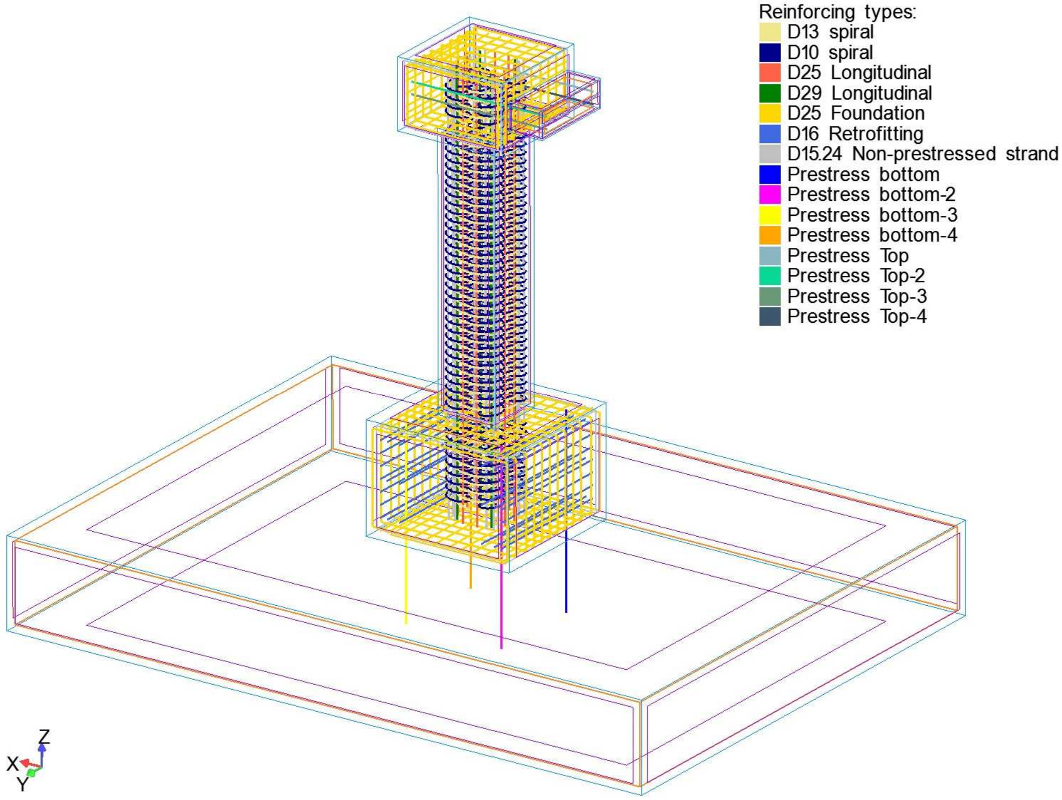


Figure 5: Wireframe model with reinforcement location

2.1.2 Materials

This section describes the materials used in this example.

Table 1: Used materials

Name	Prototype	Generator type	Base parameters	Geometry IDs
Elastic_3D	Elastic	Main	E=2000 GPa, μ=0.3, ρ=0.0025 kton/m³, α=1.2E-05 °C⁻¹	1
D13 spiral	CCReinforcement	EC2	E=180 GPa, ρ=0.00785 kton/m³, α=1.2E-05 °C⁻¹	19
Concrete for all volumes	CC3DNonLinCementitious2	EuroCode2	E=35 GPa, μ=0.2, ρ=0.0023 kton/m³, α=1.2E-05 °C⁻¹	21
D10 spiral	CCReinforcement	EC2	E=145 GPa, ρ=0.00785 kton/m³, α=1.2E-05 °C⁻¹	22
D25 Longitudinal	CCReinforcement	EC2	E=190 GPa, ρ=0.00785 kton/m³, α=1.2E-05 °C⁻¹	23
D29 Longitudinal	CCReinforcement	EC2	E=170 GPa, ρ=0.00785 kton/m³, α=1.2E-05 °C⁻¹	24
D25 Foundation	CCReinforcement	EC2	E=190 GPa, ρ=0.00785 kton/m³, α=1.2E-05 °C⁻¹	25
D16 Retrofitting	CCReinforcement	EC2	E=200 GPa, ρ=0.00785 kton/m³, α=1.2E-05 °C⁻¹	26
D15.24 Non-prestressed strand	CCReinforcement	EC2	E=195 GPa, ρ=7850 kg/m³, α=1.2E-05 °C⁻¹	27

Prestress bottom	CCReinforcement	EC2	E=195 GPa, ρ=7850 kg/m³, α=1.2E-05 °C⁻¹	29
Prestress bottom-2	CCReinforcement	EC2	E=195 GPa, ρ=7850 kg/m³, α=1.2E-05 °C⁻¹	31
Prestress bottom-3	CCReinforcement	EC2	E=195 GPa, ρ=7850 kg/m³, α=1.2E-05 °C⁻¹	32
Prestress bottom-4	CCReinforcement	EC2	E=195 GPa, ρ=7850 kg/m³, α=1.2E-05 °C⁻¹	33
Prestress Top	CCReinforcement	EC2	E=195 GPa, ρ=7850 kg/m³, α=1.2E-05 °C⁻¹	34
Prestress Top-2	CCReinforcement	EC2	E=195 GPa, ρ=7850 kg/m³, α=1.2E-05 °C⁻¹	35
Prestress Top-3	CCReinforcement	EC2	E=195 GPa, ρ=7850 kg/m³, α=1.2E-05 °C⁻¹	36
Prestress Top-4	CCReinforcement	EC2	E=195 GPa, ρ=7850 kg/m³, α=1.2E-05 °C⁻¹	37

2.1.3 Load cases

This example contains 5 load cases.

Table 2: Used load cases

Name	Load type	Load category	Multiplier	Geometry IDs
Supports	General	Undefined	1	1
Displacement	General	Undefined	1	31
Weight	General	Undefined	1	59
ReinfInitialStrain	General	Undefined	1	62
Prestress	General	Undefined	1	66
Fixed contact	General	Undefined	1	70, 71, 72, 73, 74, 75, 76, 77

2.1.4 Boundary conditions

This example contains 13 boundary conditions of these types:

- Constraint
- Displacement
- Weight
- Reinf initial strain
- Prestressing
- Fixed contact

Table 3: Used boundary conditions

Name	Type	Load case	Subject type	Geometry IDs
Constraint	Constraint	1 - Supports	Surface	24
Displacement	Displacement	2 - Displacement	Curve	1179
Weight	Weight	3 - Weight	Solid	1, 2, 3
ReinfInitialStrain	ReinfInitialStrain	4 - Reinf initial strain	Reinforcement	1159, 1160, 1161, 1162
Prestressing	Prestressing	5 - Prestress	Reinforcement	1145, 1146, 1147, 1148
FixedContact 1	FixedContact	6 - Fixed contact	Surface	1
FixedContact 6	FixedContact	6 - Fixed contact	Surface	6
FixedContact 7	FixedContact	6 - Fixed contact	Surface	7
FixedContact 8	FixedContact	6 - Fixed contact	Surface	8
FixedContact 17	FixedContact	6 - Fixed contact	Surface	17

FixedContact 167	FixedContact	6 - Fixed contact	Surface	19
FixedContact 14	FixedContact	6 - Fixed contact	Surface	14
FixedContact 175	FixedContact	6 - Fixed contact	Surface	26

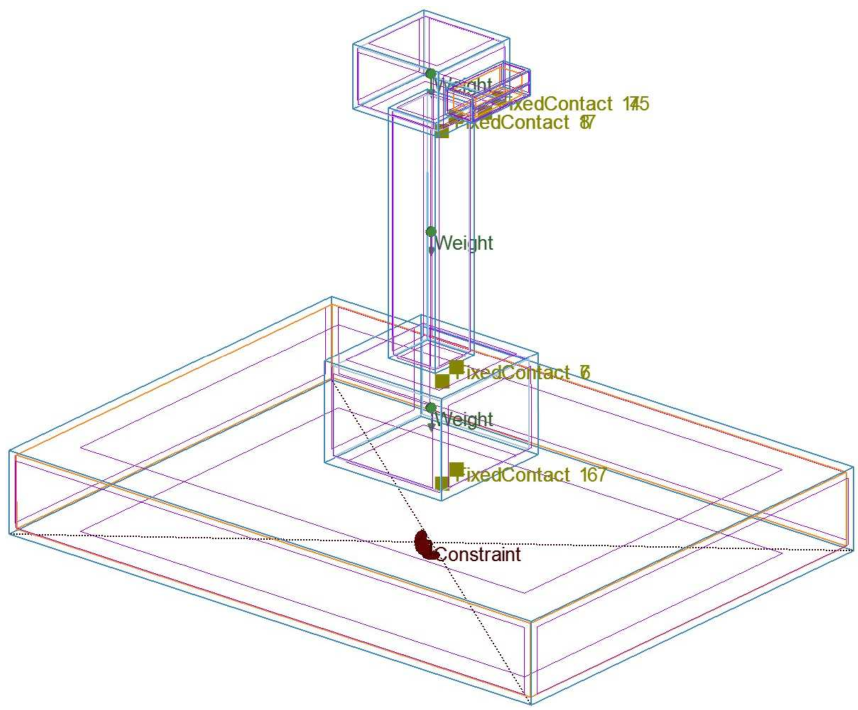


Figure 6: Boundary conditions in Interval 1

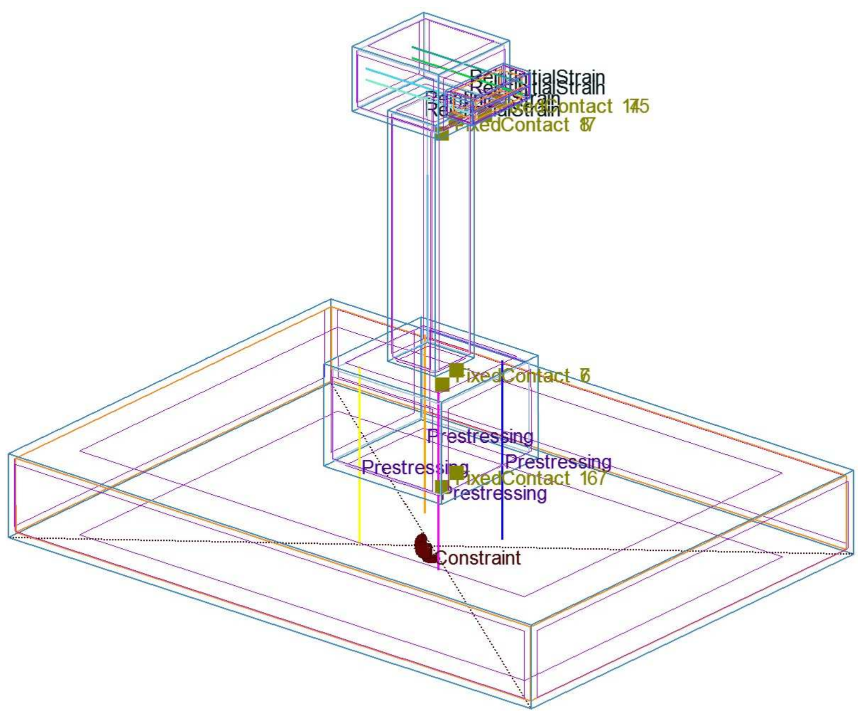


Figure 7: Boundary conditions in Interval 2

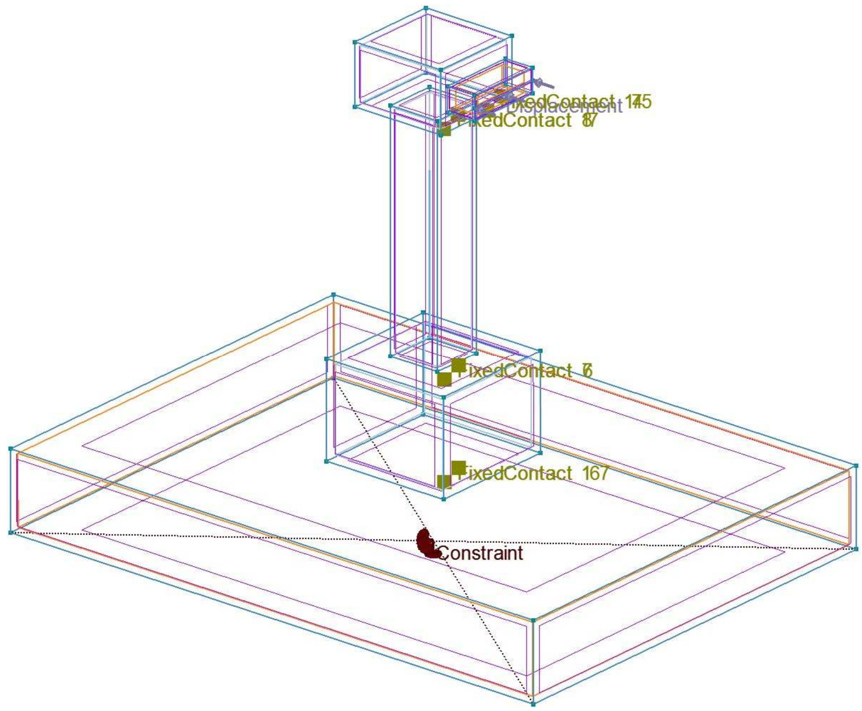


Figure 8: Boundary conditions in Interval 3

The boundary conditions in the remaining intervals (4-30) are the same as in interval 3.

2.1.5 Task definition

This example contains 30 intervals.

Table 4: List of intervals

Interval	Name	Applied load cases	Steps
1	Self weight	Supports, Weights, Fixed contact	1
2	Prestress	Supports, Prestressing, ReinfInitialStrain, Fixed contact	1
3	Displ. +0,25	Supports, Displacement, Fixed contact	5
4	Displ. -0,25	Supports, Displacement, Fixed contact	10
5	Displ. +0,375	Supports, Displacement, Fixed contact	10
6	Displ. -0,375	Supports, Displacement, Fixed contact	10
7	Displ. +0,5	Supports, Displacement, Fixed contact	10
8	Displ. -0,5	Supports, Displacement, Fixed contact	10
9	Displ. +0,75	Supports, Displacement, Fixed contact	10
10	Displ. -0,75	Supports, Displacement, Fixed contact	10
11	Displ. +1,0	Supports, Displacement, Fixed contact	30
12	Displ. -1,0	Supports, Displacement, Fixed contact	30
13	Displ. +1,5	Supports, Displacement, Fixed contact	30
14	Displ. -1,5	Supports, Displacement, Fixed contact	30
15	Displ. +2,0	Supports, Displacement, Fixed contact	30
16	Displ. -2,0	Supports, Displacement, Fixed contact	30
17	Displ. +3,0	Supports, Displacement, Fixed contact	30
18	Displ. -3,0	Supports, Displacement, Fixed contact	30
19	Displ. +4,0	Supports, Displacement, Fixed contact	30

20	Displ. -4,0	Supports, Displacement, Fixed contact	30
21	Displ. +5,0	Supports, Displacement, Fixed contact	30
22	Displ. -5,0	Supports, Displacement, Fixed contact	30
23	Displ. +6,0	Supports, Displacement, Fixed contact	30
24	Displ. -6,0	Supports, Displacement, Fixed contact	30
25	Displ. +7,0	Supports, Displacement, Fixed contact	30
26	Displ. -7,0	Supports, Displacement, Fixed contact	30
27	Displ. +8,0	Supports, Displacement, Fixed contact	30
28	Displ. -8,0	Supports, Displacement, Fixed contact	30
29	Displ. +8,0	Supports, Displacement, Fixed contact	30
30	Displ. -8,0	Supports, Displacement, Fixed contact	30

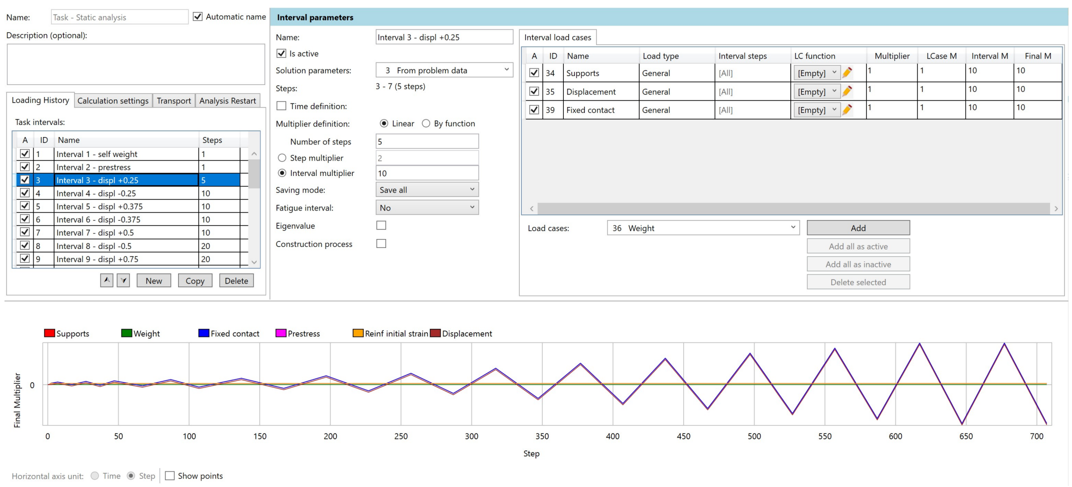


Figure 9: Task dialog

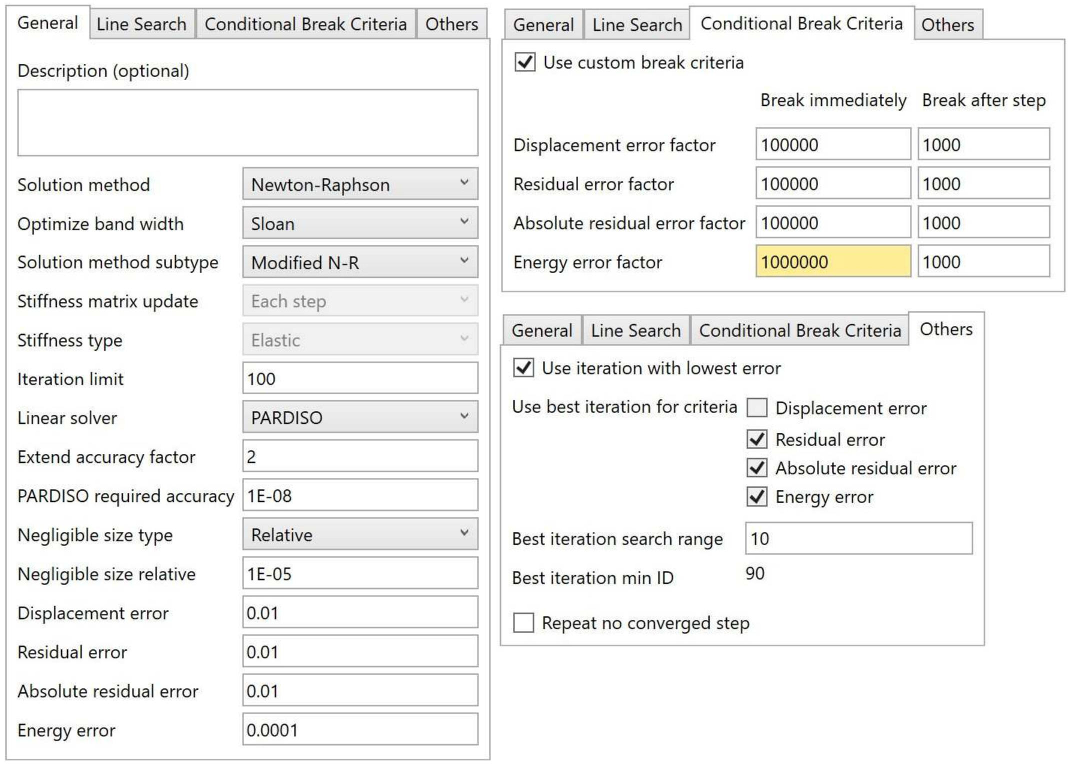


Figure 10: Solution parameters dialog

2.2 Bond Cabel Injected

Ultimate load-bearing capacity of a simple-supported prestressed concrete beam is validated in a bending test.

Initially, the prestressing stress is applied to the internal tendon and the losses of prestressing due to friction are automatically calculated in ATENA. Then, a typical loading test is performed to find the load-displacement relationship.

File name: bond_cabel_injected_left.pre

2.2.1 Geometry

Geometry is presented in the following figures (Figure 11 and Figure 12)

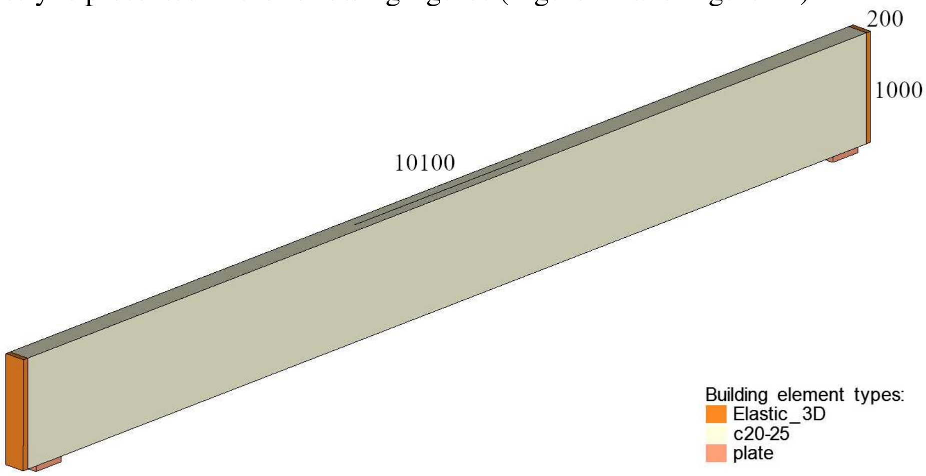


Figure 11: Solid model. Dimensions in mm

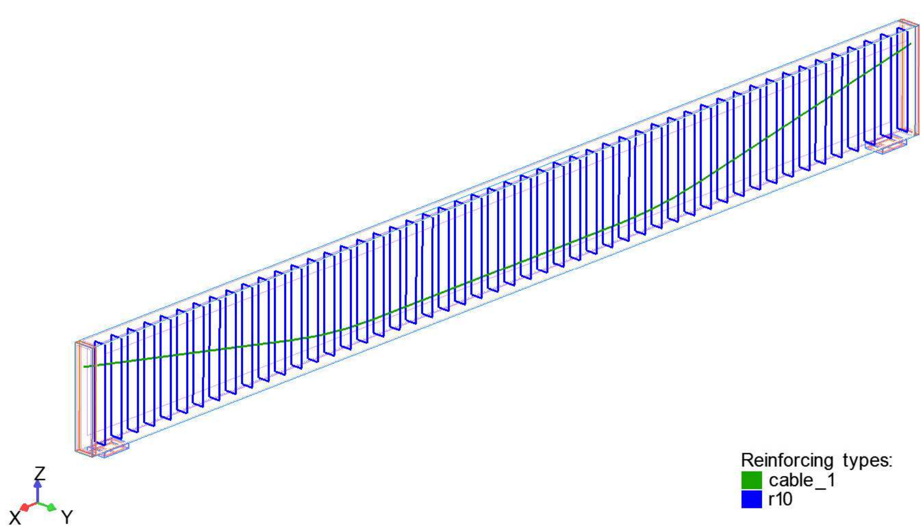


Figure 12: Wireframe model with reinforcement location

2.2.2 Materials

This section describes the materials used in this example.

Table 5: Used materials

Name	Prototype	Generator type	Base parameters	Assigned types
Elastic_3D	Elastic	Main	E=200000 MPa, μ=0.3, ρ=0.0025 kton/m³, α=1.2E-05 °C⁻¹	1, 24, 25
Reinforcement_EC2	CCReinforcement	EC2	E=200000 MPa, ρ=0.00785 kton/m³, α=1.2E-05 °C⁻¹	24, 25
cable_1	CCReinforcement	EC2	E=200 GPa, ρ=7850 kg/m³, α=1.2E-05 °C⁻¹	21
cable_2	CCReinforcement	EC2	E=200 GPa, ρ=7850 kg/m³, α=1.2E-05 °C⁻¹	22
r10	CCReinforcement	EC2	E=200000 MPa, ρ=0.00785 kton/m³, α=1.2E-05 °C⁻¹	23
c20-25	CC3DNonLinCementitious2	EuroCode2	E=30303.4 MPa, μ=0.2, ρ=0.0023 kton/m³, α=1.2E-05 °C⁻¹	24
plate	Elastic	Main	E=200000 MPa, μ=0.3, ρ=0.0025 kton/m³, α=1.2E-05 °C⁻¹	25

2.2.3 Load cases

This example contains 5 load cases.

Table 6: Used load cases

Name	Load type	Load category	Multiplier	Boundary conditions
Supports	General	Undefined	1	1, 2
Load forces	General	Undefined	1	5
Weights	General	Undefined	1	6
Prestressing	General	Undefined	1	7
Fixed contacts	General	Undefined	1	8, 9, 10

2.2.4 Boundary conditions

This example contains 8 boundary conditions of these types:

- Constraint
- Load force universaly
- Weight
- Prestressing
- Fixed contact

Table 7: Used boundary conditions

Name	Condition type	Load case	Subject type	Geometry IDs
Constraint 1	Constraint	6 - Supports	Curve	251
Constraint 2	Constraint	6 - Supports	Curve	266
LoadForceUniversally	LoadForceUniversally	7 - Load forces	Curve	23

Weight	Weight	8 - Weights	Solid	1
Prestressing	Prestressing	9 - Prestressing	Reinforcement	22
FixedContact 1	FixedContact	10 - Fixed contacts	Surface	1
FixedContact 17	FixedContact	10 - Fixed contacts	Surface	17
FixedContact 25	FixedContact	10 - Fixed contacts	Surface	24

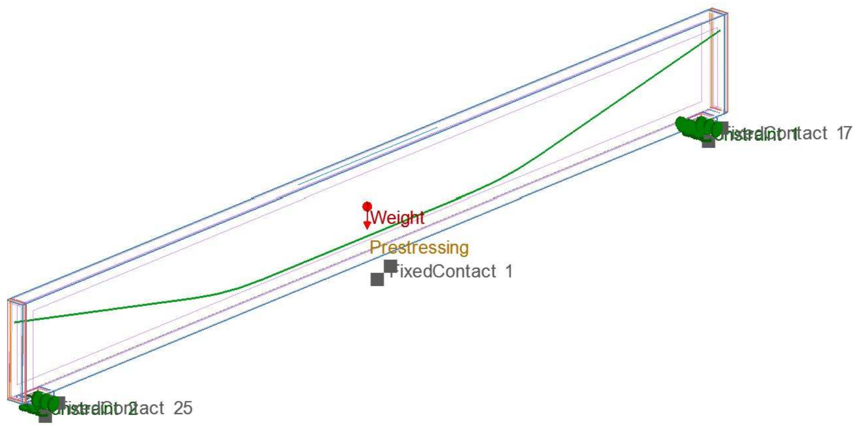


Figure 13: Boundary conditions in Interval 1

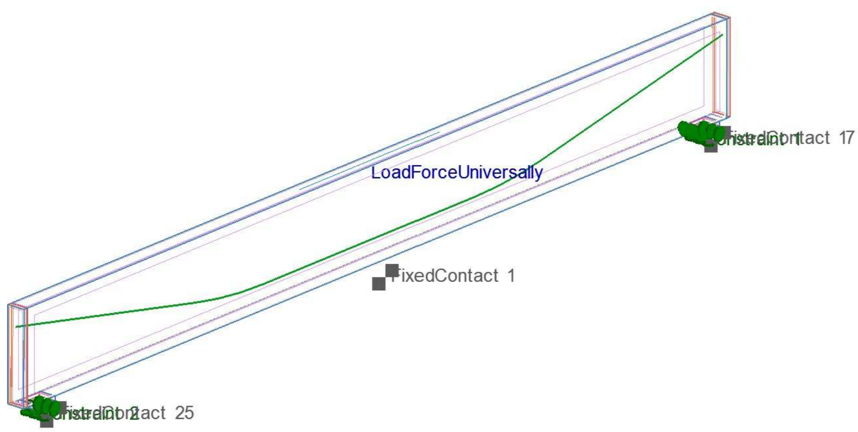


Figure 14: Boundary conditions in Interval 2

2.2.5 Task definition

This example contains 2 intervals:

Table 8: List of intervals

Interval	Name	LC combination	Steps
1	Prestressing	Supports, Weights, Prestressing, Fixed contacts	2
2	Load	Supports, Load forces, Fixed contacts	100

2.3 Shear Beam

The example shows and an analysis of a 4-point bending test with symmetry boundary conditions applied in the midspan to simplified the calculation. ATENA results predict a brittle shear failure due to lack of vertical stirrups.

File name: shear_beam.pre

2.3.1 Geometry

Geometry is presented in the following figures (Figure 15 and Figure 16)

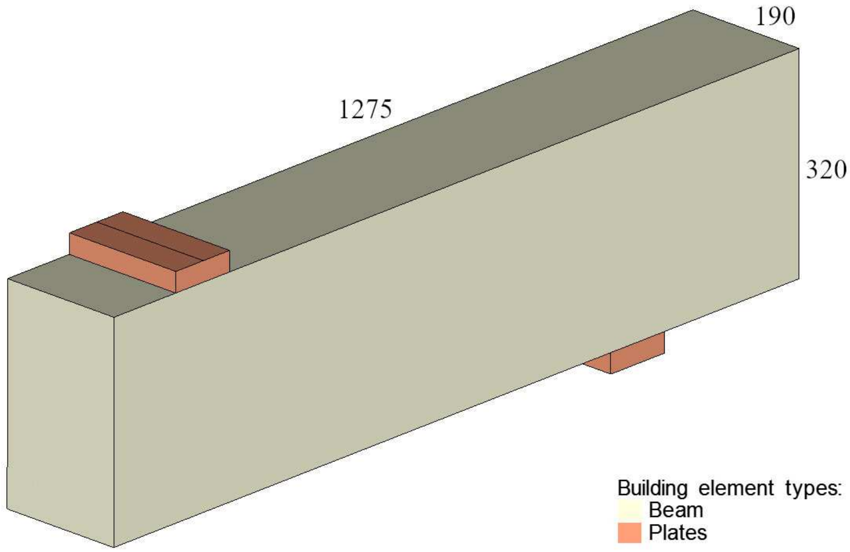


Figure 15: Solid model. Dimensions in mm

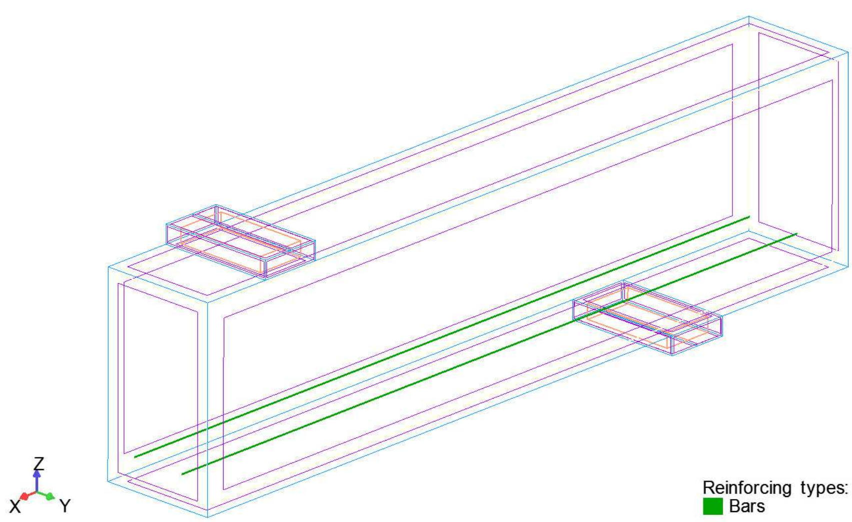


Figure 16: Wireframe model with reinforcement location

2.3.2 Materials

This section describes the materials used in this example.

Table 9: Used materials

Name	Prototype	Generator type	Base parameters	Assigned types
Elastic_3D	Elastic	Main	E=200000 MPa, μ=0.3, ρ=0.0025 kton/m³, α=1.2E-05 °C⁻¹	21, 22
Reinforcement_EC2	CCReinforcement	EC2	E=200000 MPa, ρ=0.00785 kton/m³, α=1.2E-05 °C⁻¹	21, 22
Beam	CC3DNonLinCementitious2	EuroCode2	E=31720 MPa, μ=0.2, ρ=0.0023 kton/m³, α=1.2E-05 °C⁻¹	21
Plates	Elastic	Main	E=200000 MPa, μ=0.3, ρ=0.0025 kton/m³, α=1.2E-05 °C⁻¹	22
Bars	CCReinforcement	EC2	E=208 GPa, ρ=7850 kg/m³, α=1.2E-05 °C⁻¹	23

2.3.3 Load cases

This example contains 3 load cases.

Table 10: Used load cases

Name	Load type	Load category	Multiplier	Boundary conditions
Supports	General	Undefined	1	1, 2
Displacement	General	Undefined	1	3
Fixed contacts	General	Undefined	1	4, 5, 6, 7

2.3.4 Boundary conditions

This example contains 7 boundary conditions of these types:

- Constraint
- Displacement
- Fixed contact

Table 11: Used boundary conditions

Name	Condition type	Load case	Subject type	Geometry IDs
Constraint 1	Constraint	5 - Supports	Curve	38
Constraint 2	Constraint	5 - Supports	Surface	3;;
Displacement	Displacement	6 - Displacement	Point	33
FixedContact 1	FixedContact	7 - Fixed contacts	Surface	6
FixedContact 2	FixedContact	7 - Fixed contacts	Surface	7
FixedContact 3	FixedContact	7 - Fixed contacts	Surface	16
FixedContact 4	FixedContact	7 - Fixed contacts	Surface	1

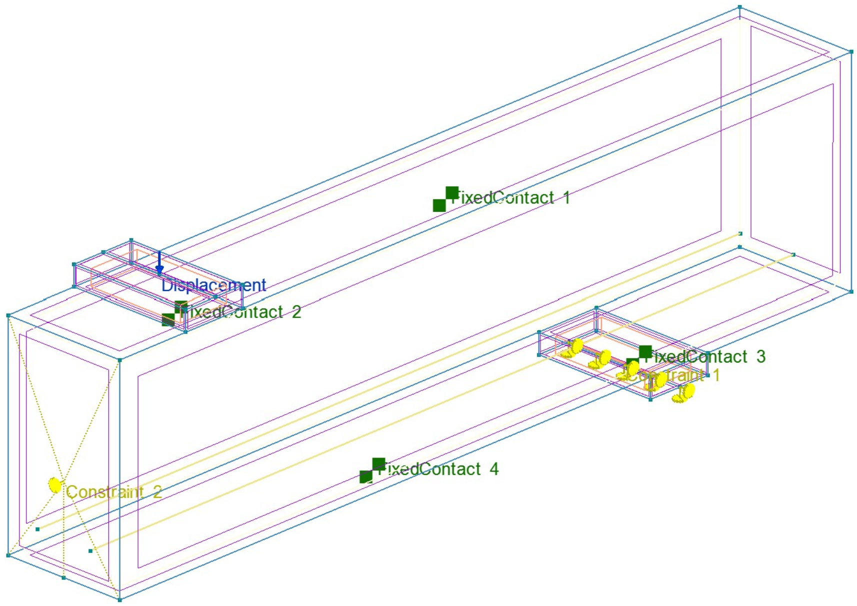


Figure 17: Boundary conditions in Interval 1

2.3.5 Task definition

This example contains only 1 interval:

Table 12: List of intervals

Interval	Name	LC combination	Steps
1	Load	Supports, Displacement, Fixed contacts	40

2.4 Shear Beam – 1d beam

The example shows and an analysis of a 4-point bending test with symmetry boundary conditions applied in the midspan to simplified the calculation. ATENA results predict a brittle shear failure due to lack of vertical stirrups. Contrary to section 2.3, the bending bottom reinforcements is modelled by 1D beam elements to take in to account the reinforcement bending stifness.

File name: shear_beam_1dbeam.pre

2.4.1 Geometry

Geometry is presented in the following figures (Figure 18 and Figure 19)

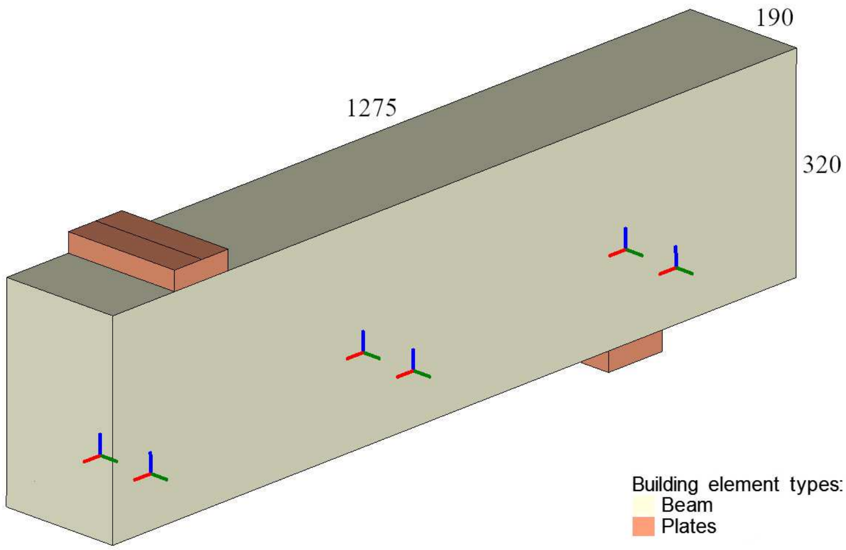


Figure 18: Solid model. Dimensions in mm

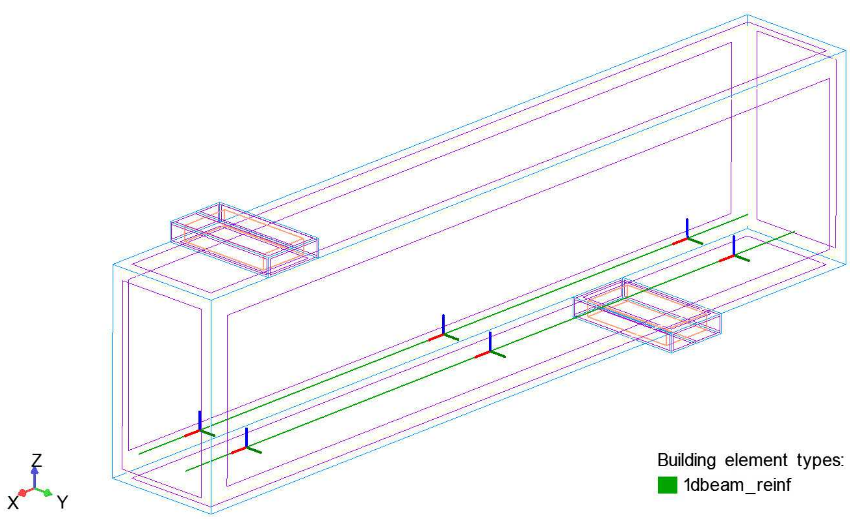


Figure 19: Wireframe model with reinforcement location

2.4.2 Materials

This section describes the materials used in this example.

Table 13: Used materials

Name	Prototype	Generator type	Base parameters	Assigned types
Elastic_3D	Elastic	Main	E=200000 MPa, μ=0.3, ρ=0.0025 kton/m³, α=1.2E-05 °C⁻¹	21, 22, 28
Reinforcement_EC2	CCReinforcement	EC2	E=200000 MPa, ρ=0.00785 kton/m³, α=1.2E-05 °C⁻¹	21, 22, 28
Beam	CC3DNonLinCementitious2	EuroCode2	E=31720 MPa, μ=0.2, ρ=0.0023 kton/m³, α=1.2E-05 °C⁻¹	21
Plates	Elastic	Main	E=200000 MPa, μ=0.3, ρ=0.0025 kton/m³, α=1.2E-05 °C⁻¹	22
Reinf	BiLinearSteelVonMises	EuroCode2	E=208 GPa, μ=0.3, ρ=0.00785 kton/m³, α=1.2E-05 °C⁻¹	28

2.4.3 Load cases

This example contains 3 load cases.

Table 14: Used load cases

Name	Load type	Load category	Multiplies	Boundary conditions
Supports	General	Undefined	1	1, 2, 3, 4, 5, 6
Displacement	General	Undefined	1	7
Fixed contacts	General	Undefined	1	8, 9, 10, 11, 12, 13

2.4.4 Boundary conditions

This example contains 13 boundary conditions of these types:

- Constraint
- Rotation constraint
- Displacement
- Fixed contact

Table 15: Used boundary conditions

Name	Condition type	Load case	Subject type	Geometry IDs
Constraint 1	Constraint	5 - Supports	Point	26
Constraint 2	Constraint	5 - Supports	Point	28
Constraint 3	Constraint	5 - Supports	Curve	38
Constraint 4	Constraint	5 - Supports	Surface	3
RotationConstraint 1	RotationConstraint	5 - Supports	Point	26
RotationConstraint 2	RotationConstraint	5 - Supports	Point	28
Displacement	Displacement	6 - Displacement	Point	33
FixedContact 1	FixedContact	7 - Fixed contacts	Curve	32
FixedContact 2	FixedContact	7 - Fixed contacts	Curve	33
FixedContact 3	FixedContact	7 - Fixed contacts	Surface	6
FixedContact 4	FixedContact	7 - Fixed contacts	Surface	7

FixedContact 5	FixedContact	7 - Fixed contacts	Surface	16
FixedContact 6	FixedContact	7 - Fixed contacts	Surface	1

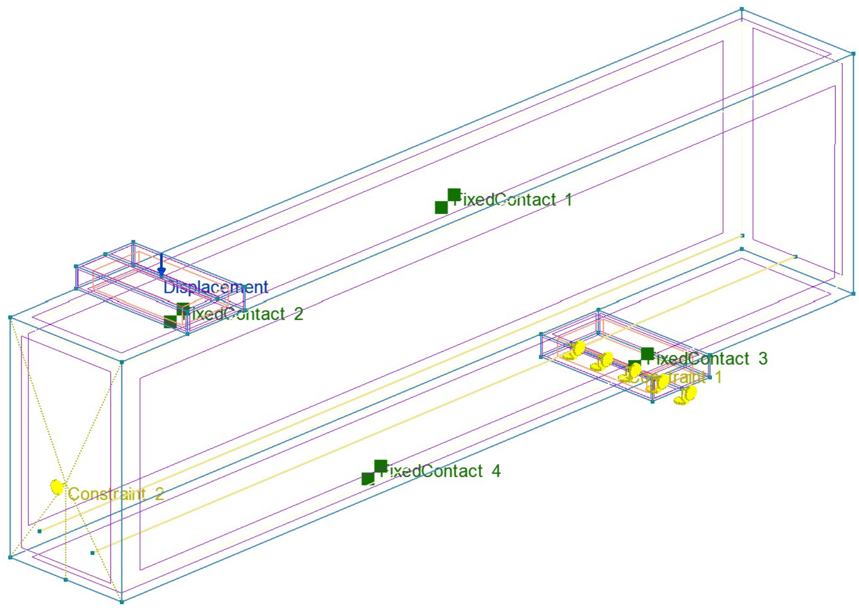


Figure 20: Boundary conditions in Interval 1

2.4.5 Task definition

This example contains only 1 interval:

Table 16: List of intervals

Interval	Name	LC combination	Steps
1	Load	Supports, Displacement, Fixed contacts	160

2.5 Interface With Shear 3D

Demonstration example of using interface modeling in ATENA.

File name: interface_with_shear3D.pre

2.5.1 Geometry

Geometry is presented in the following figures (Figure 21 and Figure 22)

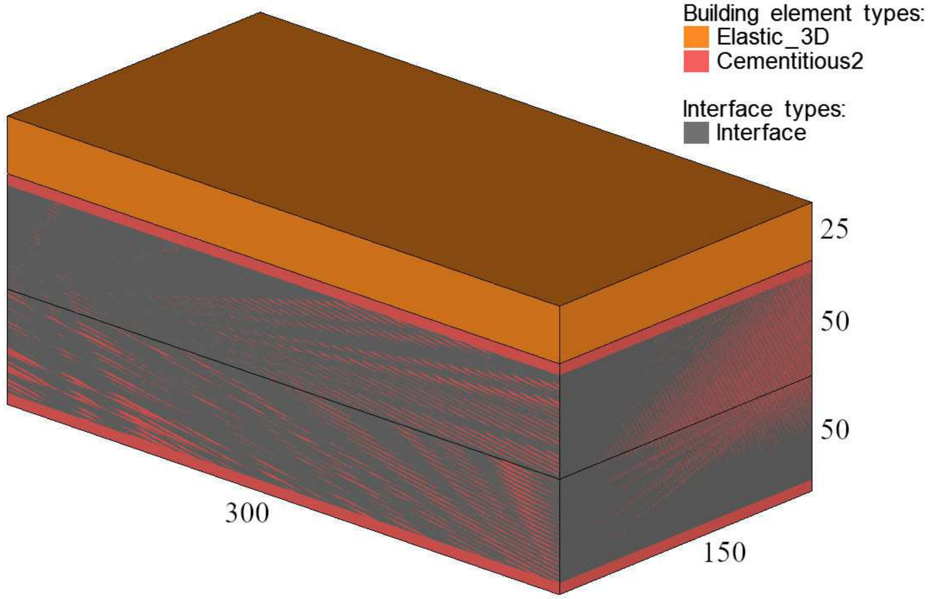


Figure 21: Solid model. Dimensions in mm

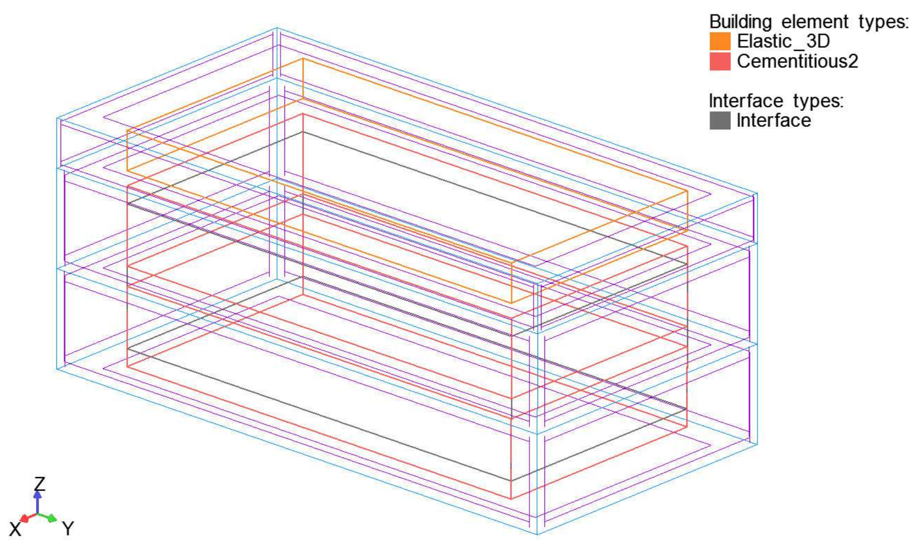


Figure 22: Wireframe model

2.5.2 Materials

This section describes the materials used in this example.

Table 17: Used materials

Name	Prototype	Generator type	Base parameters	Assigned types
Elastic_3D	Elastic	Main	E=210000 MPa, μ=0.3, ρ=0.0025 kton/m³, α=1.2E-05 °C⁻¹	1
Cementitious2	CC3DNonLinCementitious2	EuroCode2	E=30320 MPa, μ=0.2, ρ=0.0025 kton/m³, α=1.2E-05 °C⁻¹	3
Interface	Interface	Main		14

2.5.3 Load cases

This example contains 4 load cases.

Table 18: Used load cases

Name	Load type	Load category	Multiplier	Boundary conditions
Supports	General	Undefined	1	1, 2, 3, 4
Top plate support	General	Undefined	1	7
Shear displacement	General	Undefined	1	10
Initial vertical prestress	General	Undefined	1	11

2.5.4 Boundary conditions

This example contains 7 boundary conditions of these types:

- Constraint
- Displacement

Table 19: Used boundary conditions

Name	Condition type	Load case	Subject type	Geometry IDs
Constraint 1	Constraint	6 - Supports	Surface	2, 3, 14
Constraint 2	Constraint	6 - Supports	Surface	10
Constraint 3	Constraint	7 - Top plate support	Surface	7
Displacement 2	Displacement	8 - Shear displacement	Curve	7
Displacement 1	Displacement	9 - Initial vertical prestress	Surface	7

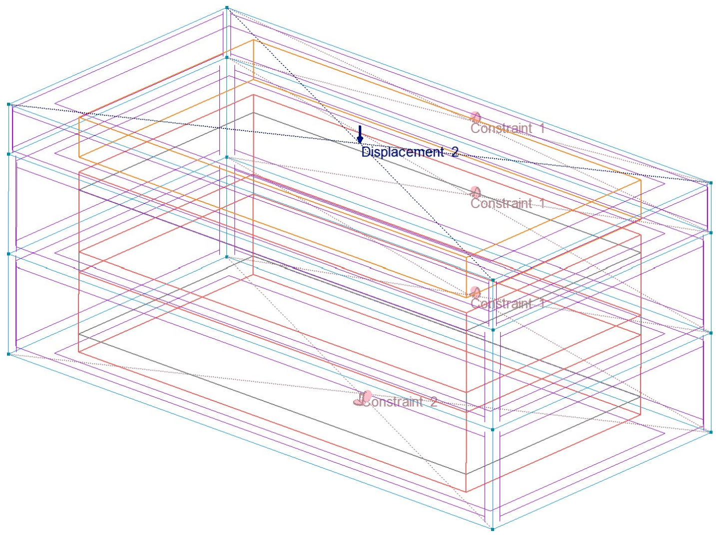


Figure 23: Boundary conditions in Interval 1

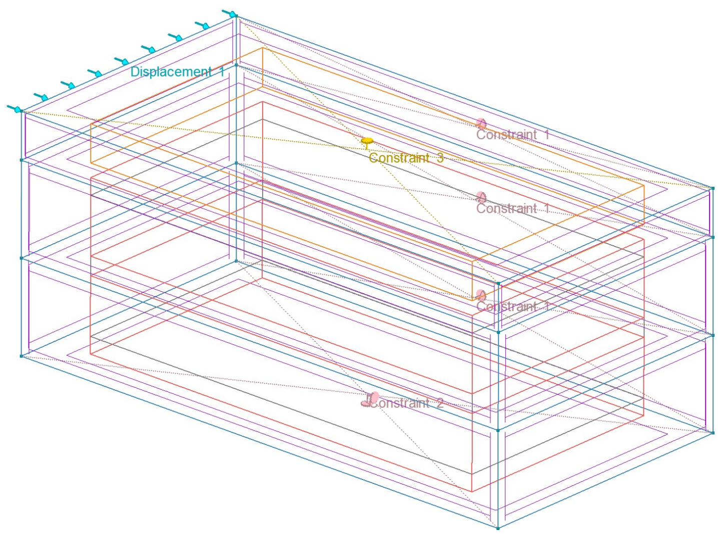


Figure 24: Boundary conditions in Interval 2

2.5.5 Task definition

This example contains 2 interval:

Table 20: List of intervals

Interval	Name	LC combination	Steps
1	Compression	Supports, Initial vertical prestress	1
2	Shear displacement	Supports, Top plate support, Shear displacement	50

2.6 Beam With Carbonation 30 years

Demonstration ATENA example for modeling carbonation of concrete.

File name: beam_with_carbonation_30years.pre

2.6.1 Geometry

Geometry is presented in the following figures (Figure 25 and Figure 26)

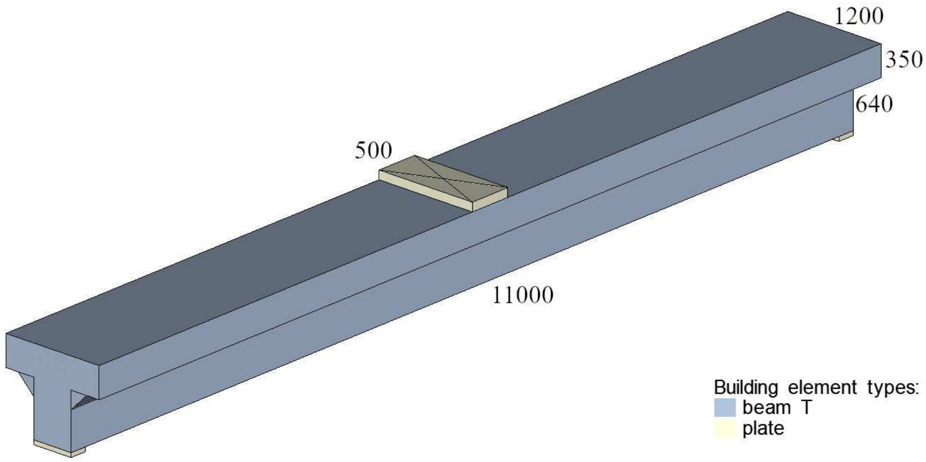


Figure 25: Solid model. Dimensions in mm

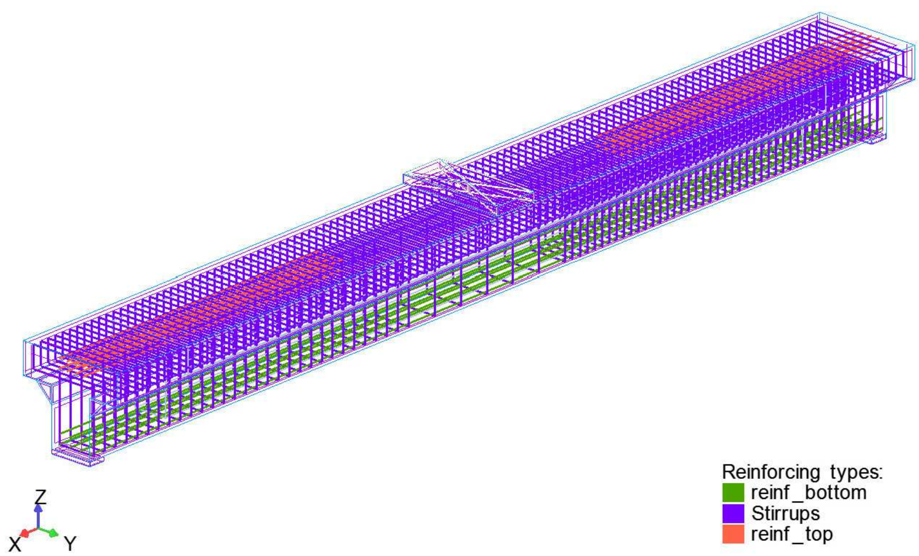


Figure 26: Wireframe model with reinforcement location

2.6.2 Materials

This section describes the materials used in this example.

Table 21: Used material

Name	Prototype	Generator type	Base parameters	Assigned types
Elastic_3D	Elastic	Main	E=200000 MPa, μ=0.3, ρ=0.0025 kton/m³, α=1.2E-05 °C⁻¹	20, 23
Reinforcement_EC2	CCReinforcement	EC2	E=200000 MPa, ρ=0.00785 kton/m³, α=1.2E-05 °C⁻¹	20, 23
beam T	CC3DNonLinCementitious2	EuroCode2	E=34000 MPa, μ=0.2, ρ=0.0023 kton/m³, α=1.2E-05 °C⁻¹	20
reinf_bottom	CCReinforcement	EC2	E=200000 MPa, ρ=0.00785 kton/m³, α=1.2E-05 °C⁻¹	21
Stirrups	CCReinforcement	EC2	E=200 GPa, ρ=7850 kg/m³, α=1.2E-05 °C⁻¹	22
plate	Elastic	Main	E=200000 MPa, μ=0.3, ρ=0.0025 kton/m³, α=1.2E-05 °C⁻¹	23
reinf_top	CCReinforcement	EC2	E=200000 MPa, ρ=0.00785 kton/m³, α=1.2E-05 °C⁻¹	24

2.6.3 Load cases

This example contains 6 load cases

Table 22: Used load cases

Name	Load type	Load category	Multiplier	Boundary conditions
Supports	General	Undefined	1	1, 2, 3
Load force	General	Undefined	1	4
Displacement	General	Undefined	1	5
Weight	General	Undefined	1	6
Carbonation	Corrosion	Undefined	1	9
Fixed contact	General	Undefined	1	16, 17, 18, 19, 20, 21, 22, 23

2.6.4 Boundary conditions

This example contains 15 boundary conditions of these types:

- Constraint
- Load force
- Displacement
- Weight
- Carbonation
- Fixed contact

Name	Condition type	Load case	Subject type	Geometry IDs
Constraint 1	Constraint	9 - Supports	Point	6
Constraint 2	Constraint	9 - Supports	Curve	288
Constraint 3	Constraint	9 - Supports	Curve	299
LoadForce	LoadForce	10 - Load force	Surface	30
Displacement	Displacement	11 - Displacement	Point	411

Weight	Weight	12 - Weight	Solid	1, 2, 5
Carbonation	Carbonation	13 - Carbonation	Surface	3, 4, 5, 6, 7, 26, 28
FixedContact 1	FixedContact	14 - Fixed contact	Surface	7
FixedContact 2	FixedContact	14 - Fixed contact	Surface	15
FixedContact 3	FixedContact	14 - Fixed contact	Surface	21
FixedContact 4	FixedContact	14 - Fixed contact	Surface	30
FixedContact 5	FixedContact	14 - Fixed contact	Surface	33
FixedContact 6	FixedContact	14 - Fixed contact	Surface	34
FixedContact 7	FixedContact	14 - Fixed contact	Surface	35
FixedContact 8	FixedContact	14 - Fixed contact	Surface	36

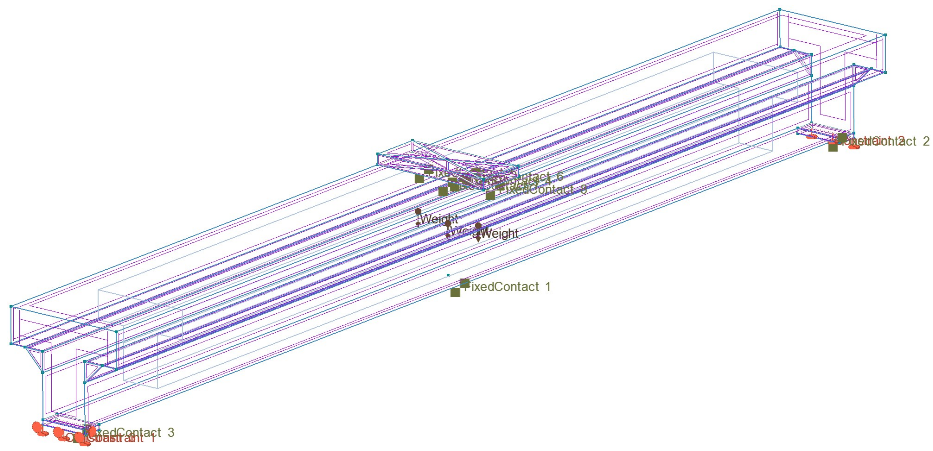


Figure 27: Boundary conditions in Interval 1

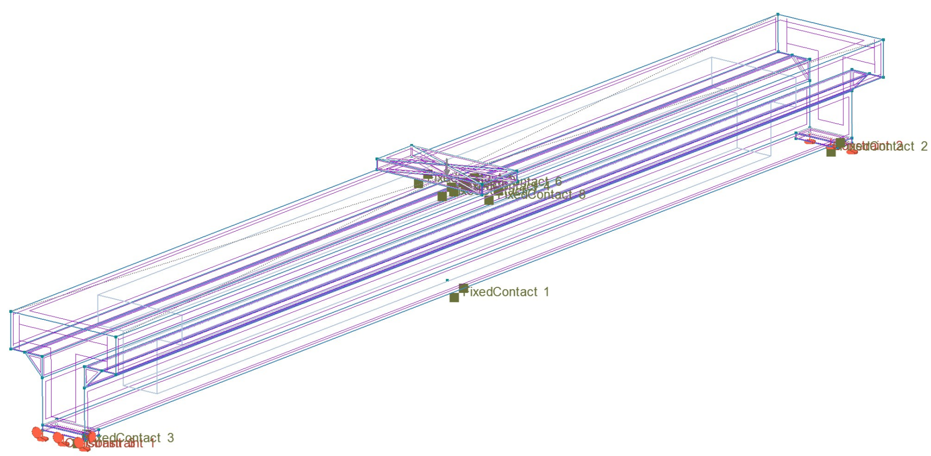


Figure 28: Boundary conditions in Interval 2

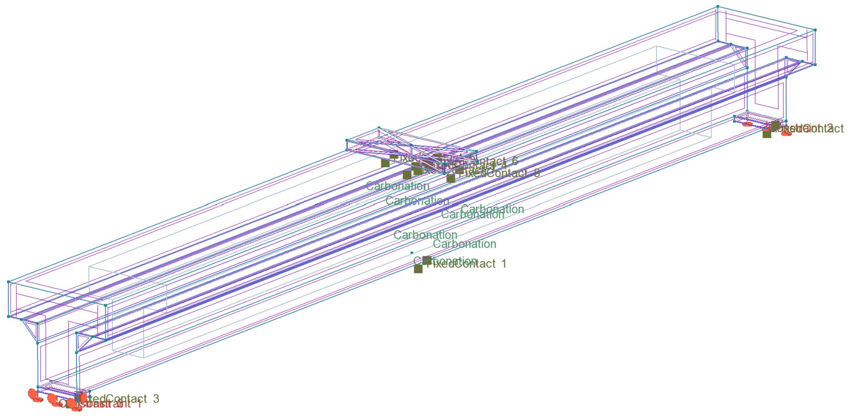


Figure 29: Boundary conditions in Interval 3

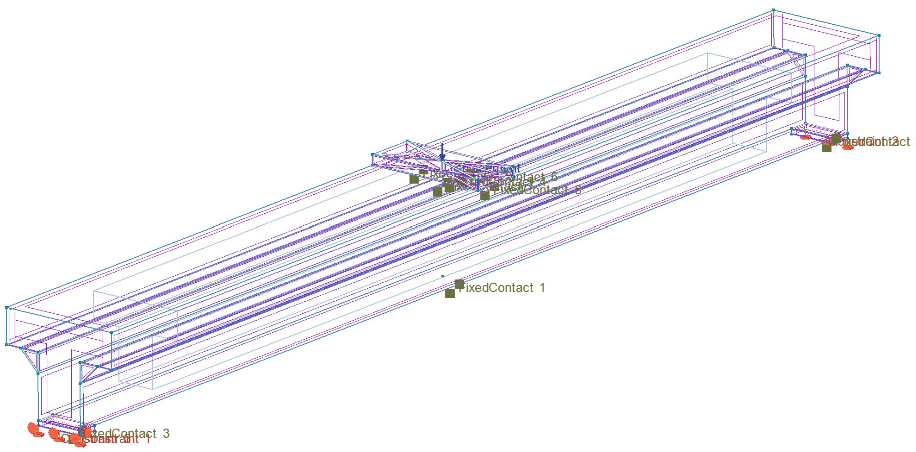


Figure 30: Boundary conditions in Interval 4

2.6.5 Task definition

This example contains 4 interval:

Table 23: List of intervals

Interval	Name	LC combination	Steps
1	Self weight	Supports, Weight, Fixed contact	10
2	Load	Supports, Load, Fixed contact	20
3	Carbonation	Supports, Carbonation, Fixed contact	20
4	Load peak	Supports, Displacement, Fixed contact	50

2.7 Beam With Chlorides 30 years

Demonstration ATENA example for modeling chloride ingress and following reinforcement corrosion in concrete.

File name: beam_with_chlorides_30years.pre

2.7.1 Geometry

Geometry is presented in the following figures (Figure 31 and Figure 32)

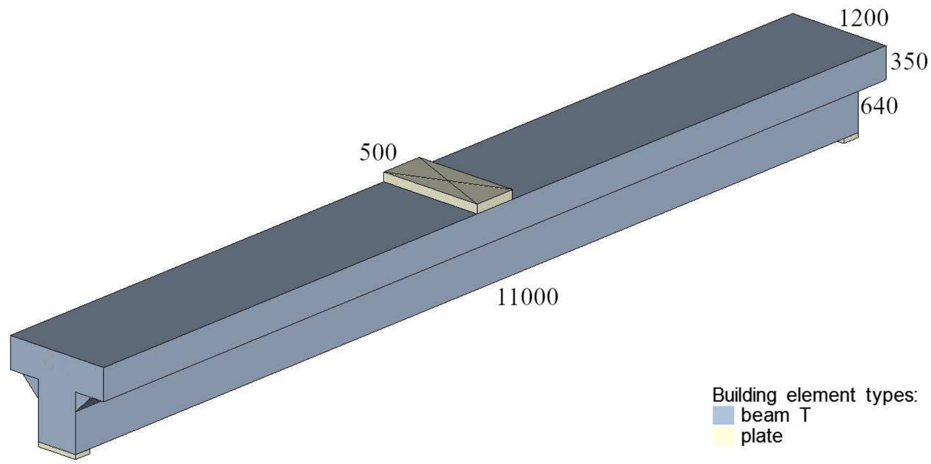


Figure 31: Solid model. Dimensions in mm

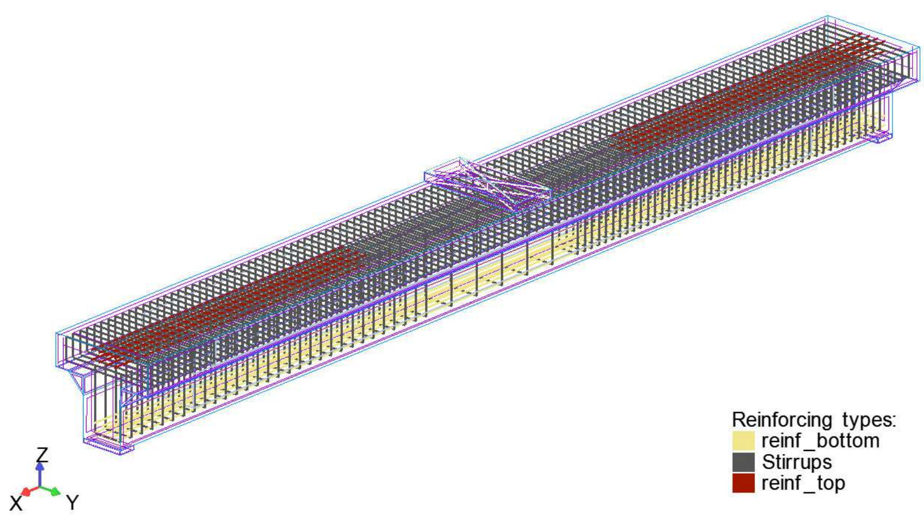


Figure 32: Wireframe model with reinforcement location

2.7.2 Materials

This section describes the materials used in this example.

Table 24: Used materials

Name	Prototype	Generator type	Base parameters	Assigned types
Elastic_3D	Elastic	Main	E=200000 MPa, μ=0.3, ρ=0.0025 kton/m³, α=1.2E-05 °C⁻¹	20, 23
Reinforcement_EC2	CCReinforcement	EC2	E=200000 MPa, ρ=0.00785 kton/m³, α=1.2E-05 °C⁻¹	20, 23
beam T	CC3DNonLinCementitious2	EuroCode2	E=34000 MPa, μ=0.2, ρ=0.0023 kton/m³, α=1.2E-05 °C⁻¹	20
reinf_bottom	CCReinforcement	EC2	E=200000 MPa, ρ=0.00785 kton/m³, α=1.2E-05 °C⁻¹	21
Stirrups	CCReinforcement	EC2	E=200 GPa, ρ=7850 kg/m³, α=1.2E-05 °C⁻¹	22
plate	Elastic	Main	E=200000 MPa, μ=0.3, ρ=0.0025 kton/m³, α=1.2E-05 °C⁻¹	23
reinf_top	CCReinforcement	EC2	E=200000 MPa, ρ=0.00785 kton/m³, α=1.2E-05 °C⁻¹	24

2.7.3 Load cases

This example contains 6 load cases.

Table 25: Used load cases

Name	Load type	Load category	Multiplier	Boundary conditions
Supports	General	Undefined	1	1, 2, 3
Load force	General	Undefined	1	4
Displacement	General	Undefined	1	5
Weights	General	Undefined	1	6
Chlorides	Corrosion	Undefined	1	9
Fixed contacts	General	Undefined	1	16, 17, 18, 19, 20, 21, 22, 23

2.7.4 Boundary conditions

This example contains 15 boundary conditions of these types:

- Constraint
- Load force
- Displacement
- Weight
- Chlorides
- Fixed contact

Table 26: Used boundary conditions

Name	Condition type	Load case	Subject type	Geometry IDs
Constraint 1	Constraint	9 - Supports	Point	6
Constraint 2	Constraint	9 - Supports	Curve	288

Constraint 3	Constraint	9 - Supports	Curve	299
LoadForce	LoadForce	10 - Load force	Surface	30
Displacement	Displacement	11 - Displacement	Point	411
Weight	Weight	12 - Weights	Solid	1, 2, 5
Chlorides	Chlorides	13 - Chlorides	Surface	3, 4, 5, 6, 7, 26, 28
FixedContact 1	FixedContact	14 - Fixed contacts	Surface	7
FixedContact 2	FixedContact	14 - Fixed contacts	Surface	15
FixedContact 3	FixedContact	14 - Fixed contacts	Surface	21
FixedContact 4	FixedContact	14 - Fixed contacts	Surface	30
FixedContact 5	FixedContact	14 - Fixed contacts	Surface	33
FixedContact 6	FixedContact	14 - Fixed contacts	Surface	34
FixedContact 7	FixedContact	14 - Fixed contacts	Surface	35
FixedContact 8	FixedContact	14 - Fixed contacts	Surface	36

2.7.5 Task definition

This example contains 4 interval:

Table 27: List of intervals

Interval	Name	LC combination	Steps
1	Self weight	Supports, Weight, Fixed contact	10
2	Load	Supports, Load, Fixed contact	20
3	Carbonation	Supports, Chlorides, Fixed contact	20
4	Load peak	Supports, Displacement, Fixed contact	50

2.8 3D Beam – strengthening B fabric 2D composite

Example for modeling of strenghtening by CFR sheets using additive CFR plates.

File name: 3Dbeam_strengthening_B_fabric_2Dcomposite.pre

2.8.1 Geometry

Geometry is presented in the following figures (Figure 33 and Figure 34)

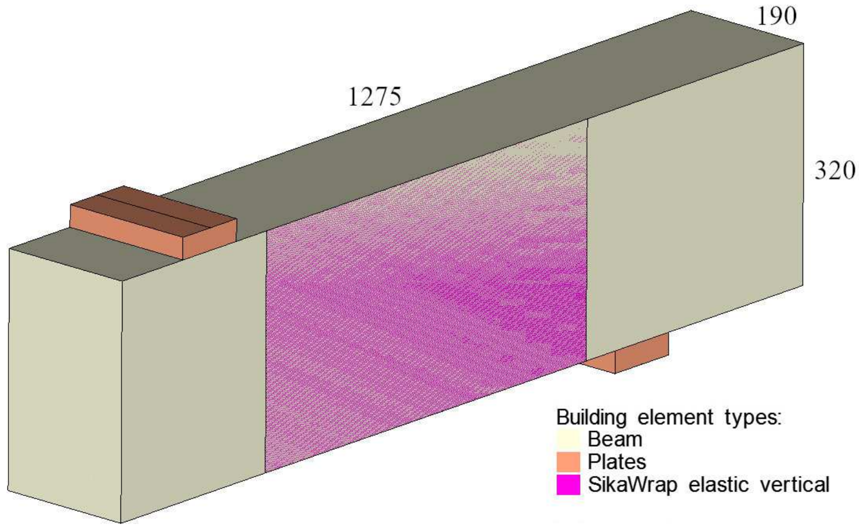


Figure 33: Solid model. Dimensions in mm

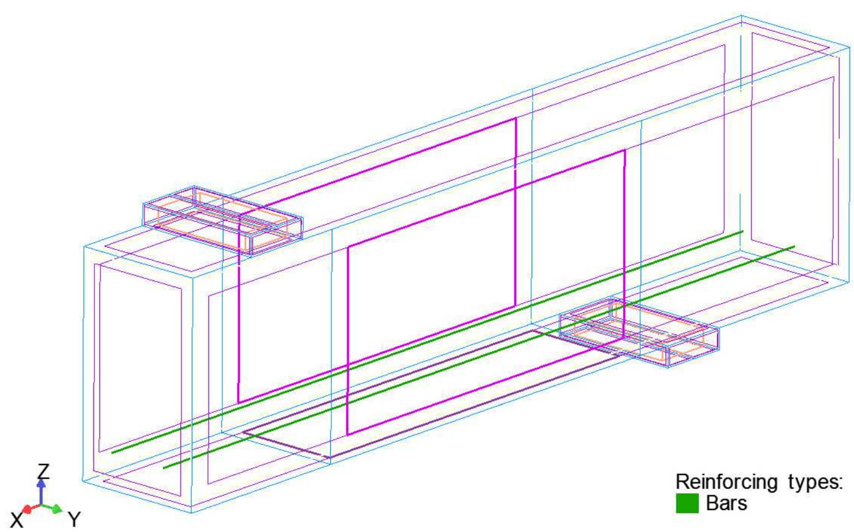


Figure 34: Wireframe model with reinforcement location

2.8.2 Materials

This section describes the materials used in this example.

Table 28: Used materials

Name	Prototype	Generator type	Base parameters	Assigned types
Beam	CC3DNonLinCementitious2	EuroCode2	E=32000 MPa, μ=0.2, ρ=0.0023 kton/m³, α=1.2E-05 °C⁻¹	21
Plates	Elastic	Main	E=200000 MPa, μ=0.3, ρ=0.0025 kton/m³, α=1.2E-05 °C⁻¹	22
Bars	CCReinforcement	EC2	E=200 GPa, ρ=7850 kg/m³, α=1.2E-05 °C⁻¹	23
WrapInterface	Interface	Main		25
SikaWrap elastic horizontal	Elastic	Main	E=2.8 MPa, μ=0.35, ρ=1310 kg/m³, α=4.5E-05 °C⁻¹	28
SikaWrap -230 C/45 vertical	CCCombinedMaterial	EuroCode2		29
SikaWrap -230 C/45 horizontal	CCCombinedMaterial	EuroCode2		30
SikaWrap elastic vertical	Elastic	Main	E=2.8 MPa, μ=0.35, ρ=1310 kg/m³, α=4.5E-05 °C⁻¹	31

2.8.3 Load cases

This example contains 3 load cases.

Table 29: Used load cases

Name	Load type	Load category	Multiplier
Supports	General	Undefined	1, 3
Displacement	General	Undefined	5
Fixed contact	General	Undefined	7, 8, 9, 10, 11, 12, 13, 14, 15

2.8.4 Boundary conditions

This example contains 12 boundary conditions of these types:

- Constraint
- Displacement
- Fixed contact

Table 30: Used boundary conditions

Name	Condition type	Load case	Subject type	Geometry IDs
Constraint 1	Constraint	6 - Supports	Curve	38
Constraint 2	Constraint	6 - Supports	Surface	3
Displacement	Displacement	7 - Displacement	Point	33
FixedContact 1	FixedContact	8 - Fixed contact	Surface	1
FixedContact 2	FixedContact	8 - Fixed contact	Surface	2
FixedContact 3	FixedContact	8 - Fixed contact	Surface	4
FixedContact 4	FixedContact	8 - Fixed contact	Surface	6
FixedContact 5	FixedContact	8 - Fixed contact	Surface	7
FixedContact 6	FixedContact	8 - Fixed contact	Surface	16

FixedContact 7	FixedContact	8 - Fixed contact	Surface	24
FixedContact 8	FixedContact	8 - Fixed contact	Surface	25
FixedContact 9	FixedContact	8 - Fixed contact	Surface	26

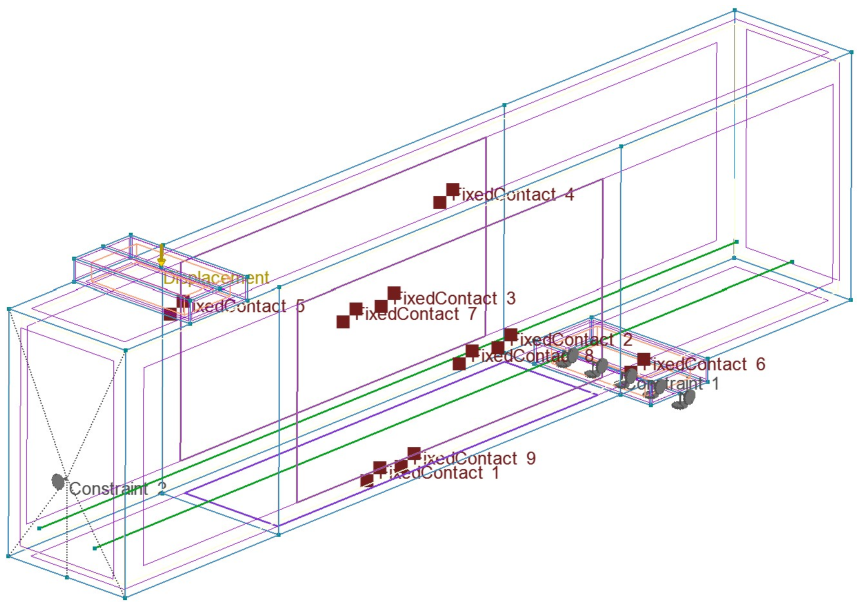


Figure 35: Boundary conditions in Interval 1

The boundary conditions in interval 2 are the same as in interval 1.

2.8.5 Task definition

This example contains 2 interval:

Table 31: List of intervals

Interval	Name	LC combination	Steps
1	Load	Supports, Displacement, Fixed contacts	1
2	Load	Supports, Displacement, Fixed contacts	50

2.9 3D Beam – strengthening B fabric 2D shell

Example for modeling of strenghtening by CFR sheets using additive CFR shell elements.

File name: 3Dbeam_ strengthening_B_fabric_2Dshell.pre

2.9.1 Geometry

Geometry is presented in the following figures (Figure 36 and Figure 37)

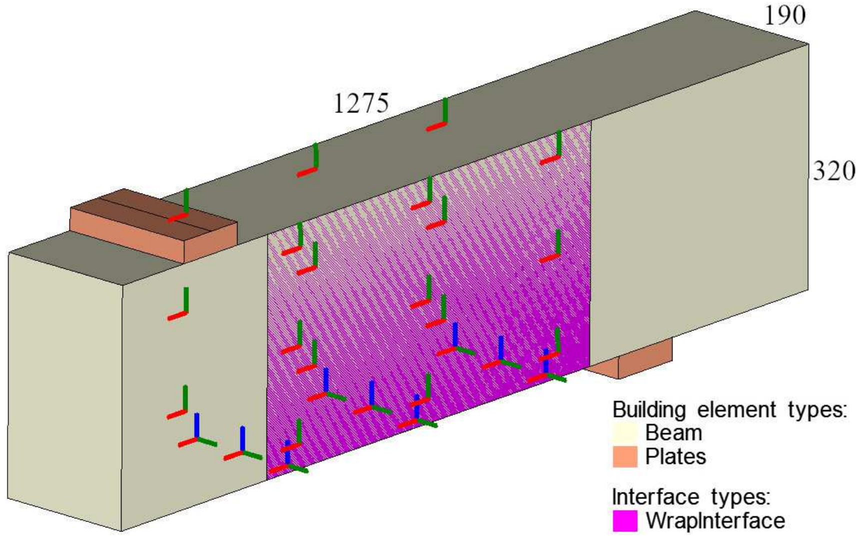


Figure 36: Solid model. Dimensions in mm

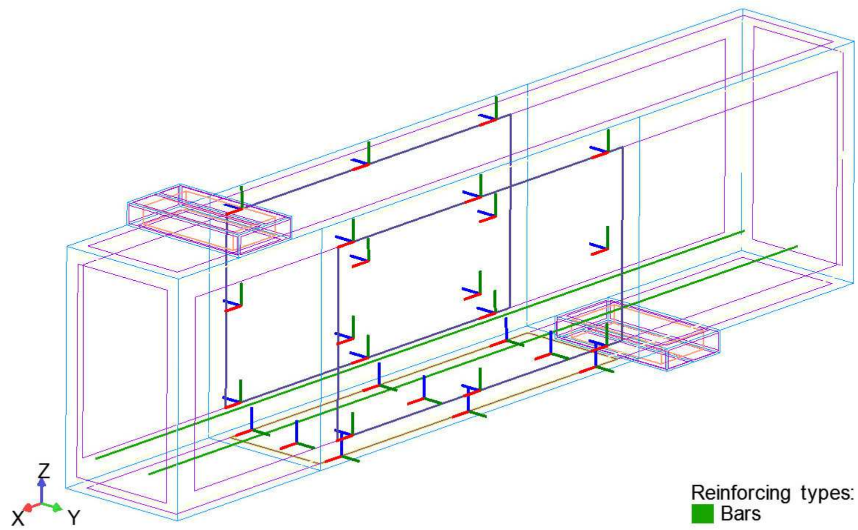


Figure 37: Wireframe model with reinforcement location

2.9.2 Materials

This section describes the materials used in this example.

Table 32: Used materials

Name	Prototype	Generator type	Base parameters	Assigned types
Beam	CC3DNonLinCementitious2	EuroCode2	E=32000 MPa, μ=0.2, ρ=0.0023 kton/m³, α=1.2E-05 °C⁻¹	21
Plates	Elastic	Main	E=200000 MPa, μ=0.3, ρ=0.0025 kton/m³, α=1.2E-05 °C⁻¹	22
Bars	CCReinforcement	EC2	E=200 GPa, ρ=7850 kg/m³, α=1.2E-05 °C⁻¹	23
WrapInterface	Interface	Main		24
SikaWrap Soft elastic	Elastic	Main	E=2.8 MPa, μ=0.35, ρ=1310 kg/m³, α=4.5E-05 °C⁻¹	32, 33
SikaWrap Base	CC3DNonLinCementitious2	EuroCode2	E=4500 MPa, μ=0.35, ρ=1310 kg/m³, α=4.5E-05 °C⁻¹	30, 31

2.9.3 Load cases

This example contains 3 load cases.

Table 33: Used load cases

Name	Load type	Load category	Multiplier
Supports	General	Undefined	1, 3
Displacement	General	Undefined	5
Fixed contact	General	Undefined	7, 8, 9, 10, 11, 12, 13, 14, 15

2.9.4 Boundary conditions

This example contains 12 boundary conditions of these types:

- Constraint
- Displacement
- Fixed contact

Table 34: Used boundary conditions

Name	Condition type	Load case	Subject type	Geometry IDs
Constraint 1	Constraint	6 - Supports	Curve	38
Constraint 2	Constraint	6 - Supports	Surface	3
Displacement	Displacement	7 - Displacement	Point	33
FixedContact 1	FixedContact	8 - Fixed contacts	Surface	1
FixedContact 2	FixedContact	8 - Fixed contacts	Surface	2
FixedContact 3	FixedContact	8 - Fixed contacts	Surface	4
FixedContact 4	FixedContact	8 - Fixed contacts	Surface	6
FixedContact 5	FixedContact	8 - Fixed contacts	Surface	7
FixedContact 6	FixedContact	8 - Fixed contacts	Surface	16
FixedContact 7	FixedContact	8 - Fixed contacts	Surface	24
FixedContact 8	FixedContact	8 - Fixed contacts	Surface	25
FixedContact 9	FixedContact	8 - Fixed contacts	Surface	26

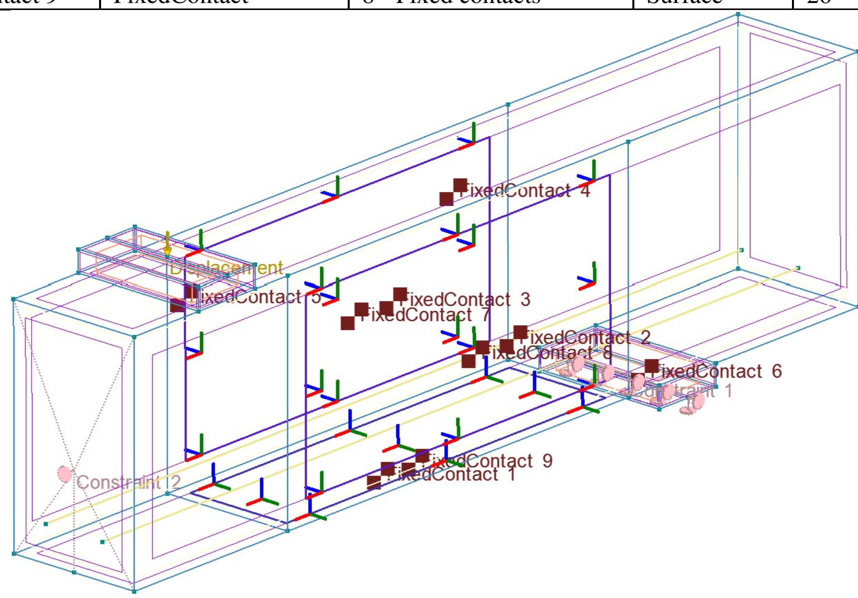


Figure 38: Boundary conditions in Interval 1

The boundary conditions in interval 2 are the same as in interval 1

2.9.5 Task definition

This example contains 2 interval:

Table 35: List of intervals

Interval	Name	LC combination	Steps
1	Load	Supports, Displacement, Fixed contacts	1

2	Load	Supports, Displacement, Fixed contacts	50

2.10 Slab With Column

This example shows modeling of connectivity between concrete slab and colummn. Typical case for slab punching.

File name: slab_with_column.pre

2.10.1 Geometry

Geometry is presented in the following figures (Figure 39 and Figure 40)

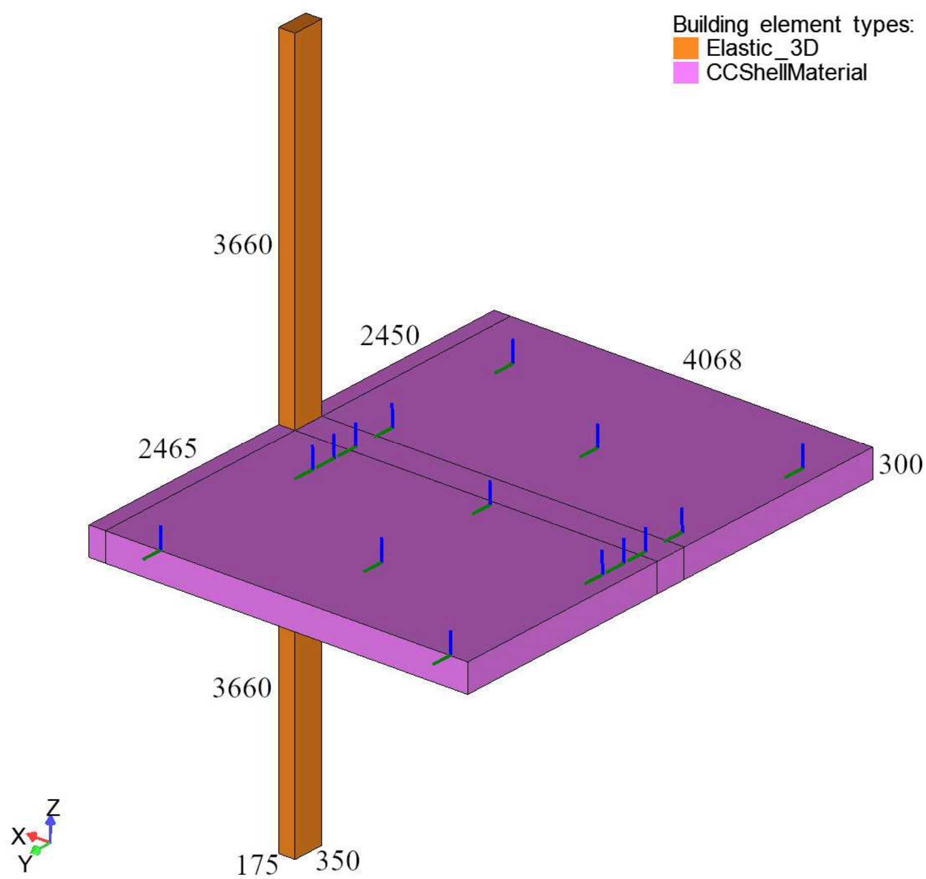


Figure 39: Solid model. Dimensions in mm

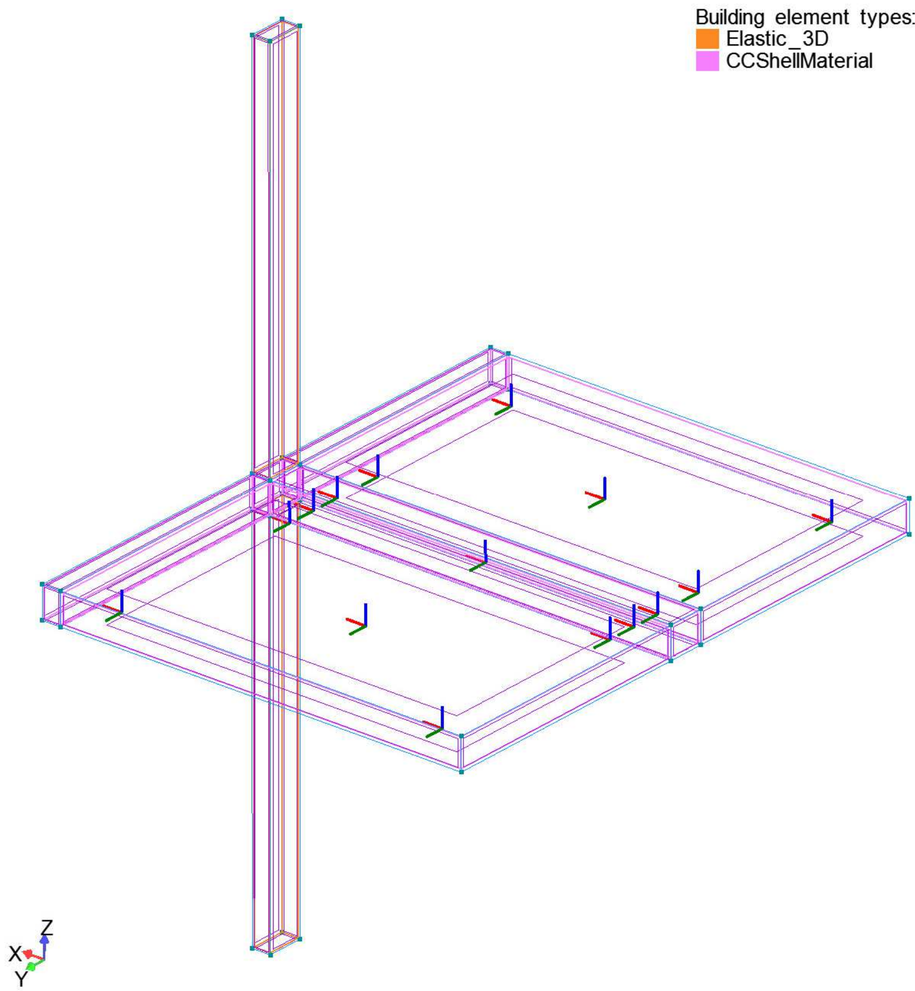


Figure 40: Wireframe model

2.10.2 Materials

This section describes the materials used in this example.

Table 36: Used materials

Name	Prototype	Generator type	Base parameters	Assigned types
Elastic_3D	Elastic	Main	E=200000 MPa, μ=0.3, ρ=0.0025 kton/m³, α=1.2E-05 °C⁻¹	1, 25
Cementitious2	CC3DNonLinCementitious2	EuroCode2	E=34000 MPa, μ=0.2, ρ=0.0025 kton/m³, α=1.2E-05 °C⁻¹	25
Reinforcement_EC2	CCReinforcement	EC2	E=200000 MPa, ρ=0.00785 kton/m³, α=1.2E-05 °C⁻¹	25

2.10.3 Load cases

This example contains 6 load cases.

Table 37: Used load cases

Name	Load type	Load category	Multiplier	Boundary conditions
Supports	General	Undefined	1	1, 13, 15, 19, 21
Load force 1	General	Undefined	1	49
Load force 2	General	Undefined	1	54
Weights	General	Undefined	1	59
Initial strain	General	Undefined	1	67
Fixed contacts	General	Undefined	1	73, 74, 75, 76

2.10.4 Boundary conditions

The following 13 boundary conditions were used in this example:

- Constraint
- Load force
- Weight
- Initial strain
- Fixed contact

Table 38: Used boundary conditions

Name	Condition type	Load case	Subject type	Geometry IDs
Constraint 1	Constraint	8 - Supports	Curve	10, 14, 17
Constraint 2	Constraint	8 - Supports	Surface	8, 12, 21, 30, 35
Constraint 3	Constraint	8 - Supports	Surface	13, 19
Constraint 4	Constraint	8 - Supports	Surface	34
Constraint 5	Constraint	8 - Supports	Surface	39
LoadForce 1	LoadForce	9 - Load force 1	Surface	24, 25, 26, 27, 28
LoadForce 2	LoadForce	10 - Load force 2	Surface	24, 25, 26, 27, 28
Weight	Weight	11 - Weights	Solid	1, 2, 3, 4, 5, 6, 7, 8
InitialStrain	InitialStrain	12 - Initial strain	Solid	1, 2, 3, 4, 5, 6
FixedContact 1	FixedContact	13 - Fixed contacts	Surface	29
FixedContact 2	FixedContact	13 - Fixed contacts	Surface	6
FixedContact 3	FixedContact	13 - Fixed contacts	Surface	40
FixedContact 4	FixedContact	13 - Fixed contacts	Surface	41

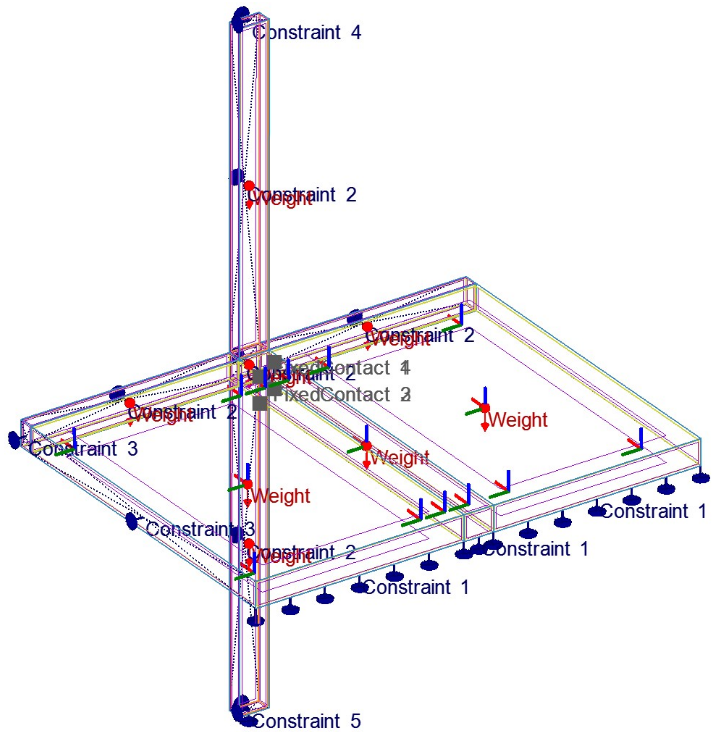


Figure 41: Boundary conditions in Interval 1

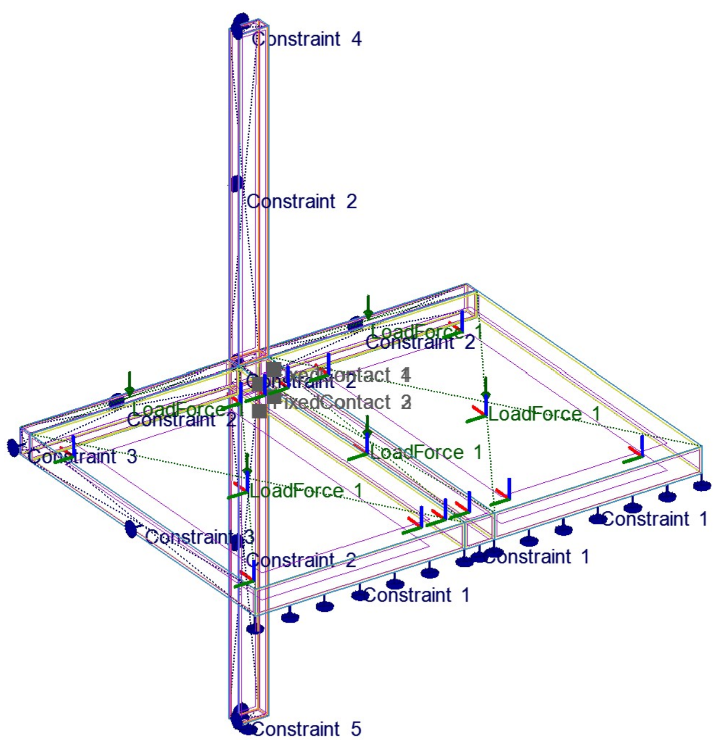


Figure 42: Boundary conditions in Interval 2

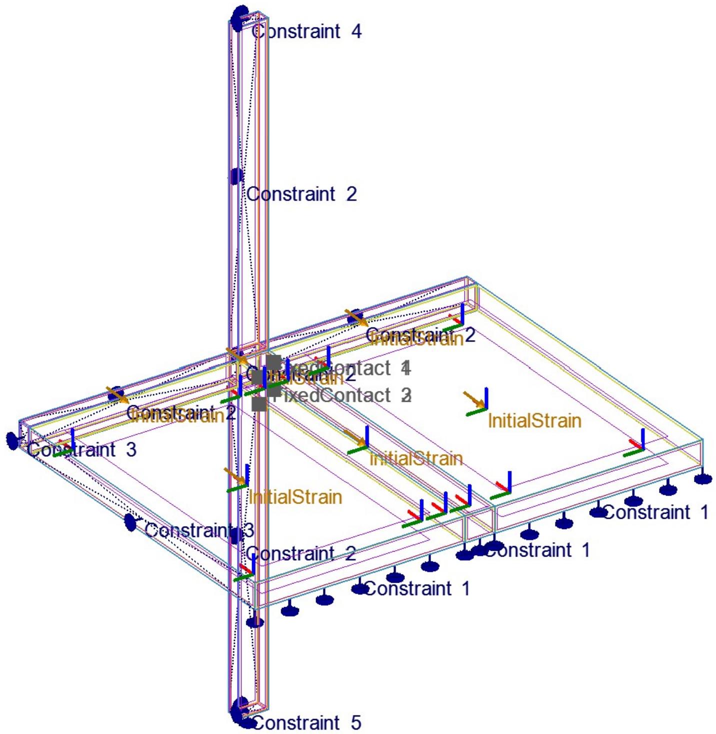


Figure 43: Boundary conditions in Interval 3

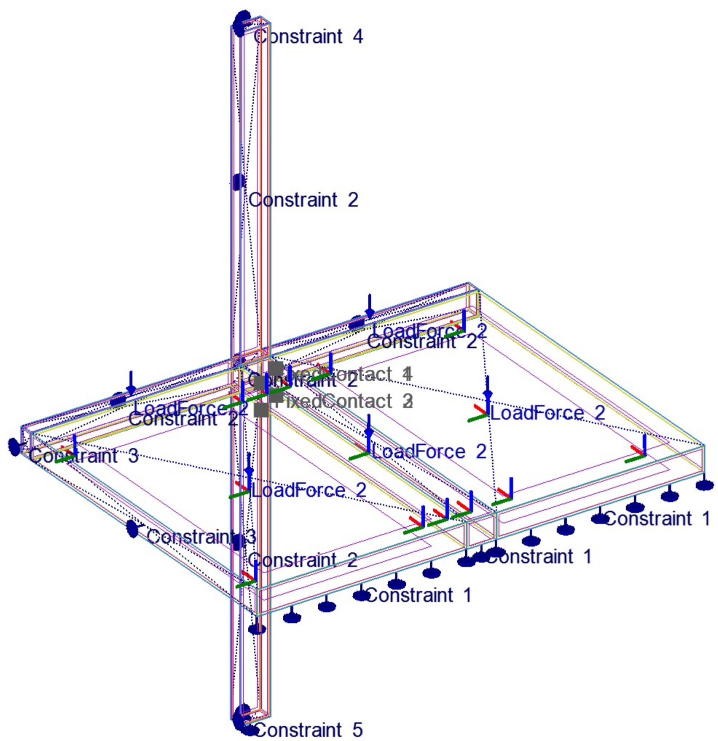


Figure 44: Boundary conditions in Interval 4

2.10.5 Task definition

This example contains 4 interval:

Table 39: List of intervals

Interval	Name	LC combination	Steps
1	Self weight	Supports, Weights, Fixed contacts	5
2	Live load	Supports, Load force 1, Fixed contacts	5
3	Shrinkage	Supports, Initial strain, Fixed contacts	10
4	Live load	Supports, Load force 2, Fixed contacts	10

3. DYNAMIC ANALYSIS

This chapter contains examples of dynamic analysis using the program **ATENA 2023 CeSTaR2**.

3.1 Single Degree Free Vibration Dynamic

Simple example for dynamic modeling in ATENA.

File name: single_degree_free_vibration_dynamic.pre

3.1.1 Geometry

Geometry is presented in the following figures (Figure 45 and Figure 46)

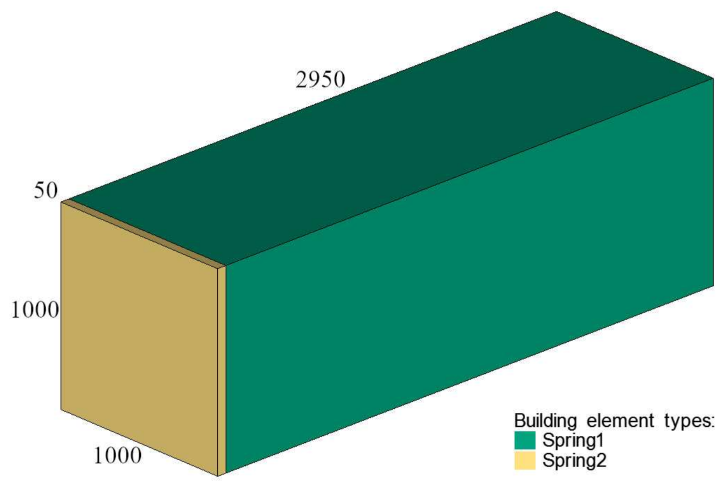


Figure 45: Solid model. Dimensions in mm

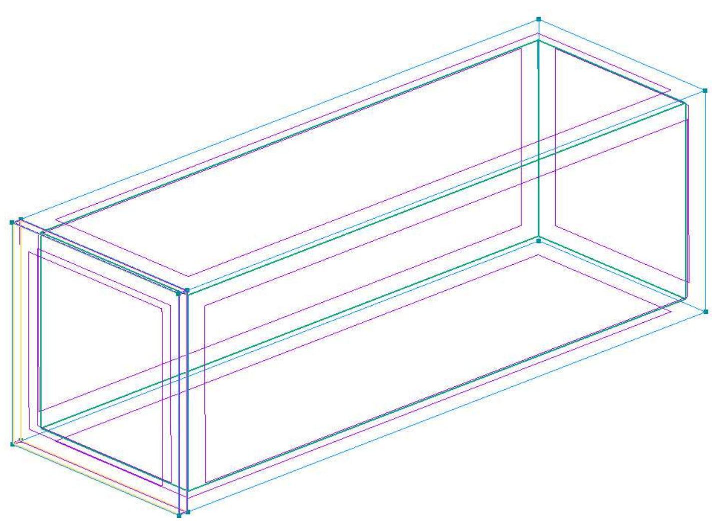


Figure 46: Wireframe model

3.1.2 Materials

This section describes the materials used in this example.

Table 40: Used materials

Name	Prototype	Generator type	Base parameters	Assigned types
Spring1	Elastic	Main	E=30 MPa, μ=0, ρ=100 kg/m³, α=1.2E-05 °C⁻¹	31
Spring2	Elastic	Main	E=300 GPa, μ=0, ρ=7850 kg/m³, α=1.2E-05 °C⁻¹	32

3.1.3 Load cases

This example contains 2 load cases.

Table 41: Used load cases

Name	Load type	Load category	Multiplier	Boundary conditions
Supports	True	General	Undefined	1, 2, 5
Displacement	True	General	Undefined	7, 8, 9, 10

3.1.4 Boundary conditions

The following 5 boundary conditions were used in this example:

- Constraint
- Displacement

Table 42: Used boundary conditions

Name	Condition type	Load case	Subject type	Geometry IDs
Constraint 1	Constraint	6 - Supports	Curve	4
Constraint 2	Constraint	6 - Supports	Curve	9
Constraint 3	Constraint	6 - Supports	Surface	5
Displacement 1	Displacement	7 - Displacement	Point	2
Displacement 2	Displacement	7 - Displacement	Point	3
Displacement 3	Displacement	7 - Displacement	Point	11
Displacement 4	Displacement	7 - Displacement	Point	12

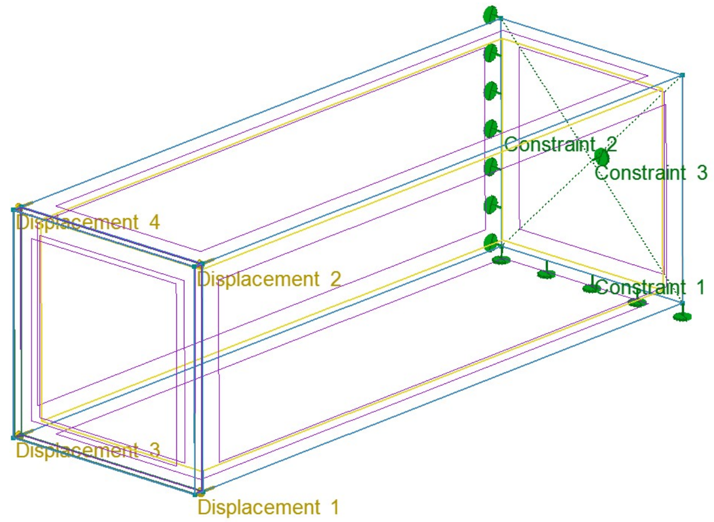


Figure 47: Boundary conditions in Interval 1

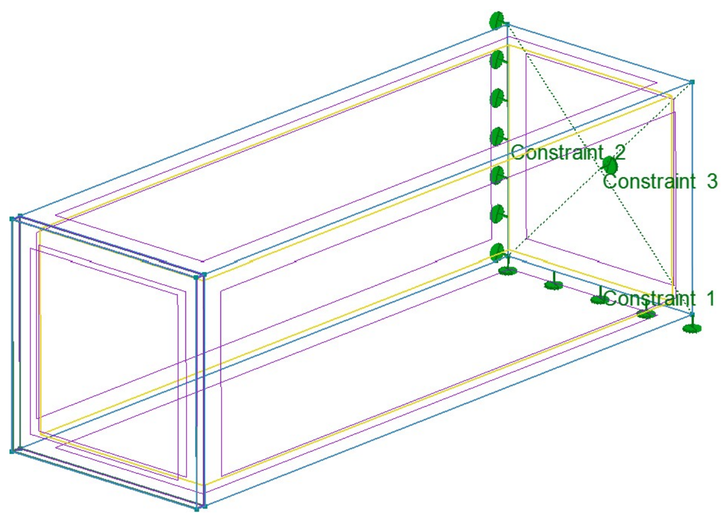


Figure 48: Boundary conditions in Interval 1

3.1.5 Task definition

This example contains 2 interval:

Table 43: List of intervals

Interval	Name	LC combination	Steps
1	Harmonic load	Supports, Displacement	8
2	Free vibration	Supports	100

3.2 CAMUS 3D Accelerogram Melendy Ranch

Complex practical example for modeling of dynamic structural response in ATENA.

File name: camus_3Daccelerogram_melendy_ranch

3.2.1 Geometry

Geometry is presented in the following figures (Figure 49 and Figure 50)

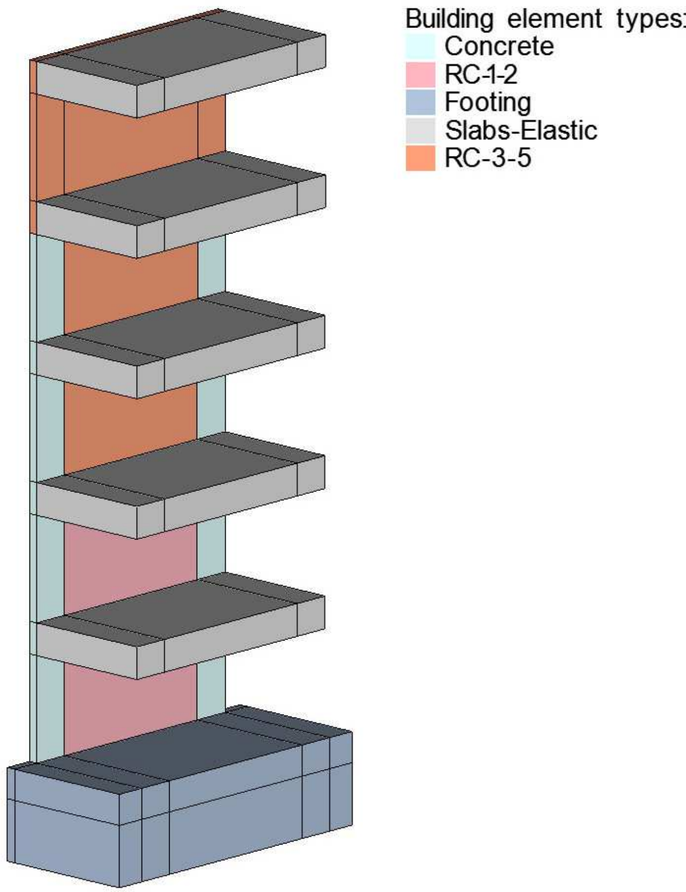


Figure 49: Solid model. Dimensions in mm

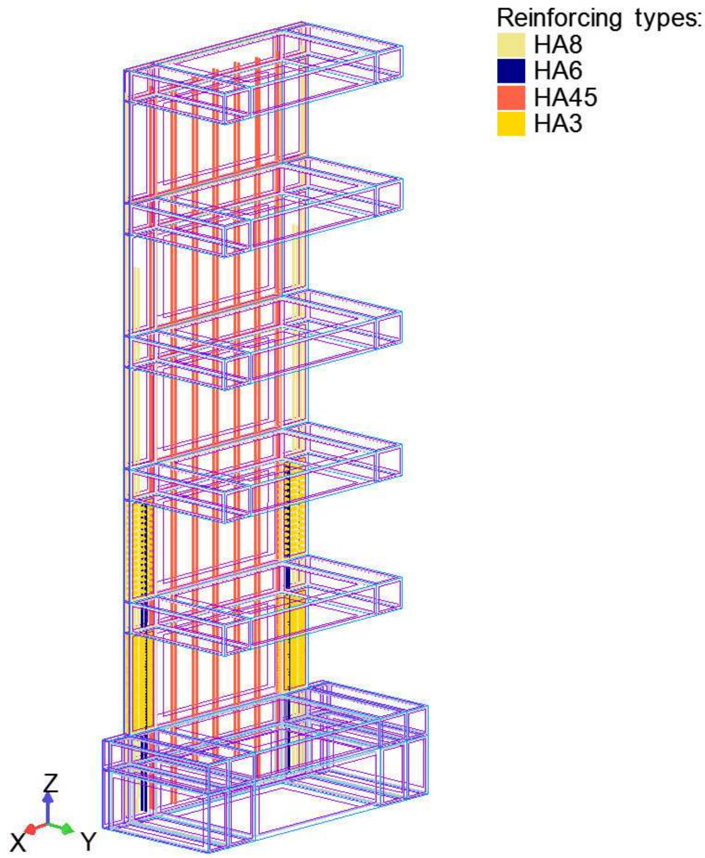


Figure 50: Wireframe model with reinforcement location

3.2.2 Materials

The materials, used in this example, are:

Table 44: Used materials

Name	Prototype	Generator type	Base parameters	Assigned types
Concrete	CC3DNonLinCementitious2	EuroCode2	E=31 GPa, μ=0.2, ρ=0.0023 kton/m³, α=1.2E-05 °C⁻¹	17
RC-1-2	CCCombinedMaterial	EuroCode2		18
Footing	Elastic	Main	E=25000 MPa, μ=0.2, ρ=0.0023 kton/m³, α=1.2E-05 °C⁻¹	20
HA8	CCReinforcement	EC2	E=200000 MPa, ρ=0 kton/m³, α=1.2E-05 °C⁻¹	21
HA6	CCReinforcement	EC2	E=200000 MPa, ρ=0 kton/m³, α=1.2E-05 °C⁻¹	22
HA45	CCReinforcement	EC2	E=200000 MPa, ρ=0 kton/m³, α=1.2E-05 °C⁻¹	23
Slabs-Elastic	Elastic	Main	E=10 GPa, μ=0.2, ρ=0.01 kton/m³, α=1.2E-05 °C⁻¹	25
HA3	CCReinforcement	EC2	E=200000 MPa, ρ=0 kton/m³, α=1.2E-05 °C⁻¹	27
RC-3-5	CCCombinedMaterial	EuroCode2		28

3.2.3 Load cases

This example contains 2 load cases.

Table 45: Used load cases

Name	Load type	Load category	Multiplier	Boundary conditions
Supports	True	General	Undefined	1, 16
Weight	True	General	Undefined	71

3.2.4 Boundary conditions

The following 2 boundary conditions were used in this example:

- Constraint
- Weight

Name	Condition type	Load case	Subjec t type	Geometry IDs
Constraint 1	Constraint	6 - Supports	Surface	40, 41, 42, 76, 77, 78, 115, 116, 117, 154, 155, 156, 193, 194, 195, 270, 271, 272, 273, 274, 275, 276, 277, 278, 279
Constraint 2	Constraint	6 - Supports	Surface	228, 229, 230, 231, 232, 265, 266, 267, 268, 269
Weight	Weight	7 - Weight	Solid	1, 2, 3, 4, 5, 6, 7, 8, 9, 10, 11, 12, 13, 14, 15, 16, 17, 18, 19, 20, 21, 22, 23, 24, 25, 26, 27, 28, 29, 30, 31, 32, 33, 34, 35, 36, 37, 38, 39, 40, 41, 42, 43, 44, 45, 46, 47, 48, 49, 50, 51, 52, 53, 54, 55, 56, 57, 58, 59, 60, 61, 62, 63, 64, 65

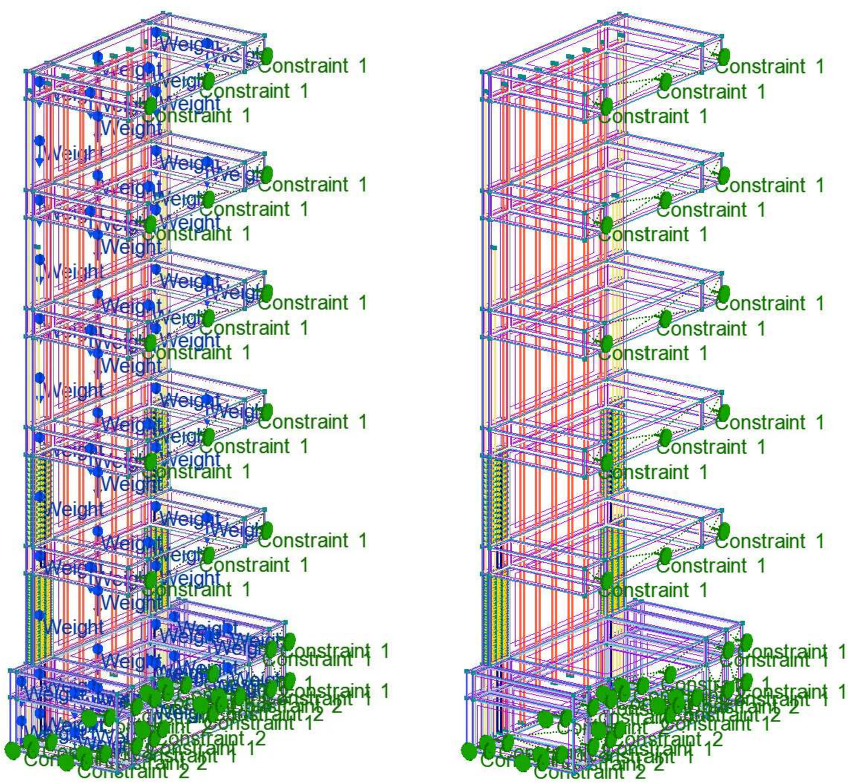


Figure 51: Boundary conditions in Interval 1 (left) and Interval 2 (right)

3.2.5 Task definition

This example contains 2 interval:

Table 46: List of intervals

Interval	Name	LC combination	Steps
1	Self weight	Supports, Weight	2
2	Earthquake	Supports	499

4. CREEP ANALYSIS

This chapter contains examples of creep analysis using the program **ATENA 2023 CeSTaR2**.

4.1 Reinforced Slab With Spring Support

Example of long term behavior of reinforced concrete slab supported by springs.

File name: reinforced_slab_with_spring_support

4.1.1 Geometry

Geometry is presented in the following figures (Figure 52 and Figure 53)

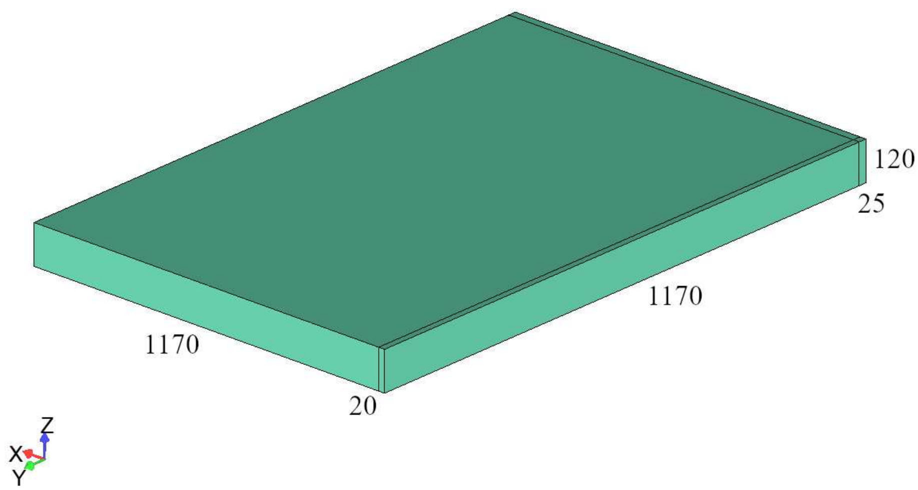


Figure 52: Solid model. Dimensions in mm

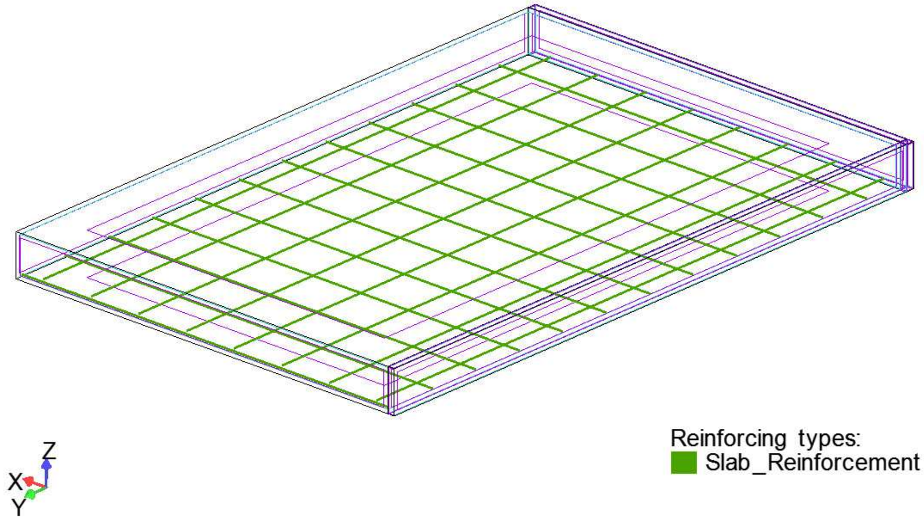


Figure 53: Wireframe model with reinforcement location

4.1.2 Materials

This section describes the materials used in this example.

Table 47: Used materials

Name	Prototype	Generator type	Base parameters	Assigned types
Slab_Reinforcement	CCReinforcement	EC2	E=210000 MPa, ρ=7850 kg/m³, α=1.2E-05 °C⁻¹	21

4.1.3 Load cases

This example contains 3 load cases.

Table 48: Used load cases

Name	Load type	Load category	Multiplier	Boundary conditions
Supports	General	Undefined	1	1, 2
Load force	General	Undefined	1	7
Weights	General	Undefined	1	8

4.1.4 Boundary conditions

The following 3 boundary conditions were used in this example:

- Constraint
- Load force
- Weight

Name	Condition type	Load case	Subject type	Geometry IDs
Constraint 1	Constraint	7 - Supports	Surface	2
Constraint 2	Constraint	7 - Supports	Surface	5

LoadForce	LoadForce	8 - Load force	Surface	6
Weight	Weight	9 - Weights	Solid	

4.1.5 Task definition

This example contains 3 interval:

Table 49: List of intervals

Interval	Name	LC combination	Steps
1	Self weight	Supports, Weights	5
2	Load forces	Supports, Load force	10
3	Creep for 450 day	Support	10

4.2 Slab With Column

Example of long term behavior of reinforced concrete slab supported by column..

File name: slab_with_column_creep.pre

4.2.1 Geometry

Geometry is presented in the following figures (Figure 54 and Figure 55)

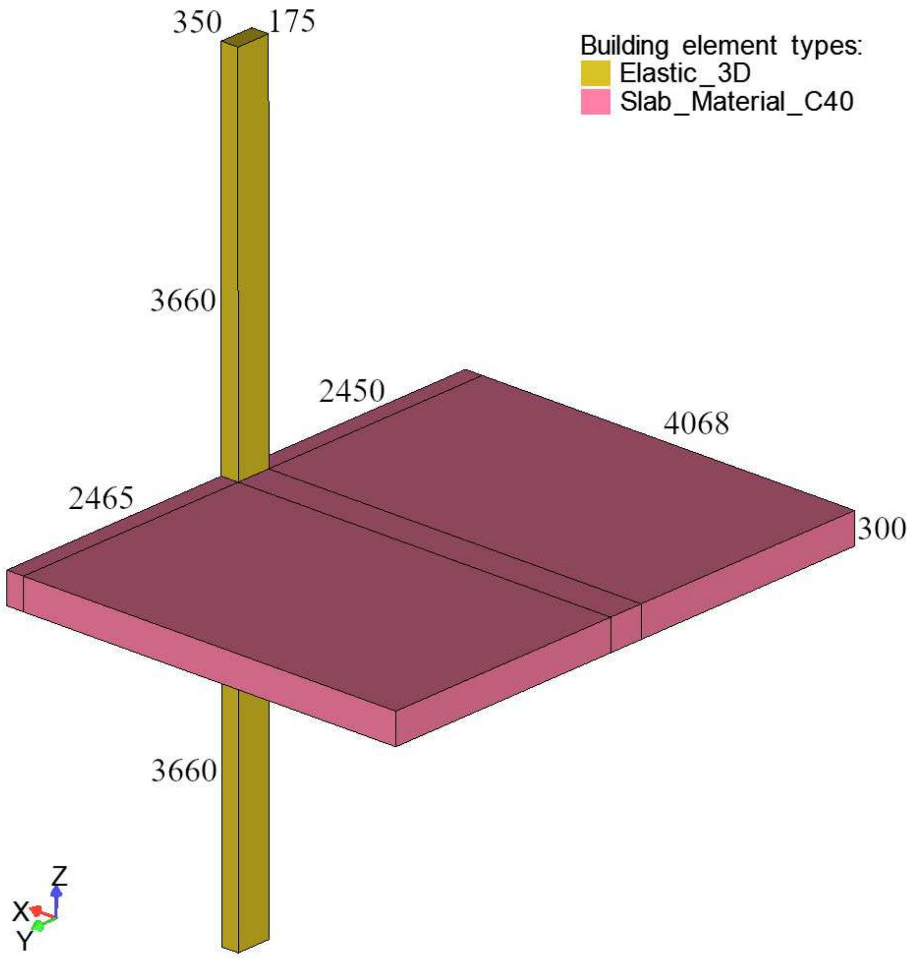


Figure 54: Solid model. Dimensions in mm

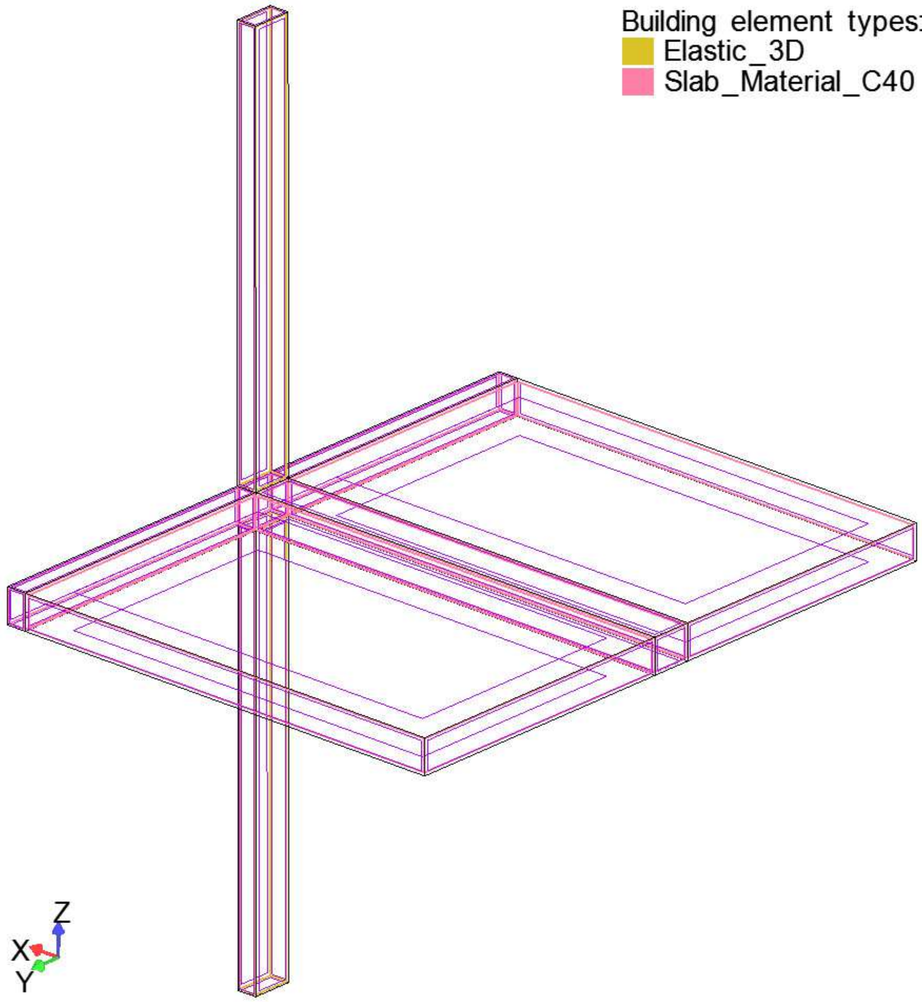


Figure 55: Wireframe model

4.2.2 Materials

This section describes the materials used in this example.

Table 50: Used materials

Name	Prototype	Generator type	Base parameters	Assigned types
Elastic_3D	Elastic	Main	E=210000 MPa, μ=0.3, ρ=0.0025 kton/m³, α=1.2E-05 °C⁻¹	2
C40_for_the_slab	CC3DNonLinCementitious2	EuroCode2	E=35000 MPa, μ=0.2, ρ=0.0023 kton/m³, α=1.2E-05 °C⁻¹	26

4.2.3 Load cases

This example contains 3 load cases.

Table 51: Used load cases

Name	Load type	Load category	Multiplier	Boundary conditions
Supports	General	Undefined	1	1, 3, 11
Weights	General	Undefined	1	30
Load force	General	Undefined	1	25

4.2.4 Boundary conditions

The following 5 boundary conditions were used in this example:

- Constraint
- Load force
- Weight

Table 52: Used boundary conditions

Name	Condition type	Load case	Subject type	Geometry IDs
Constraint 1	Constraint	6 - Supports	Surface	8, 12, 16, 20, 21, 23, 30, 35
Constraint 2	Constraint	6 - Supports	Surface	13, 19
Constraint 3	Constraint	6 - Supports	Surface	34, 39
LoadForce	LoadForce	8 - Load force	Surface	24, 25, 26, 27, 28
Weight	Weight	7 - Weights	Solid	1, 2, 3, 4, 5, 6, 7, 8

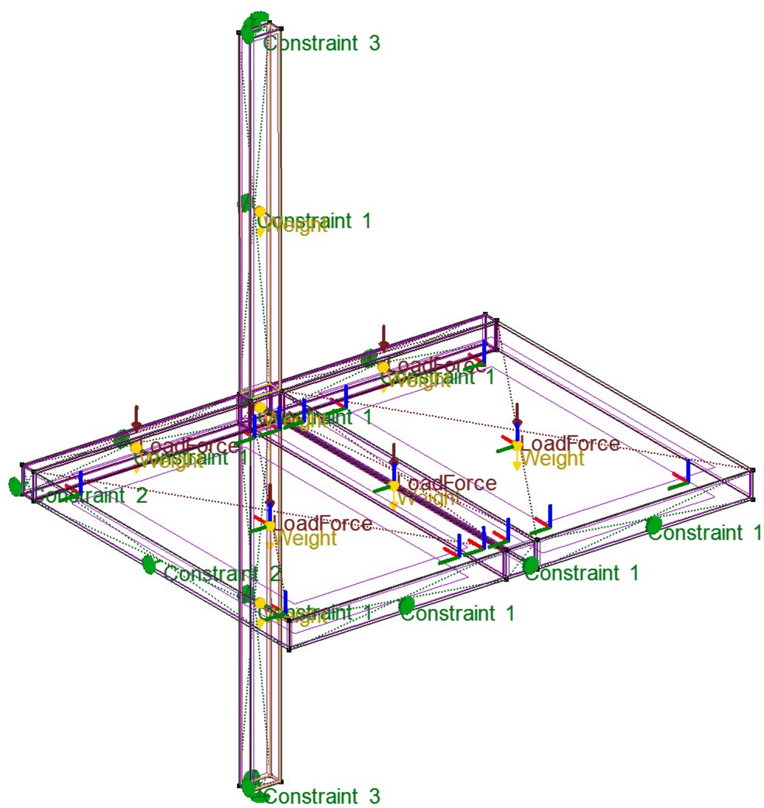


Figure 56: Boundary conditions in Interval 1

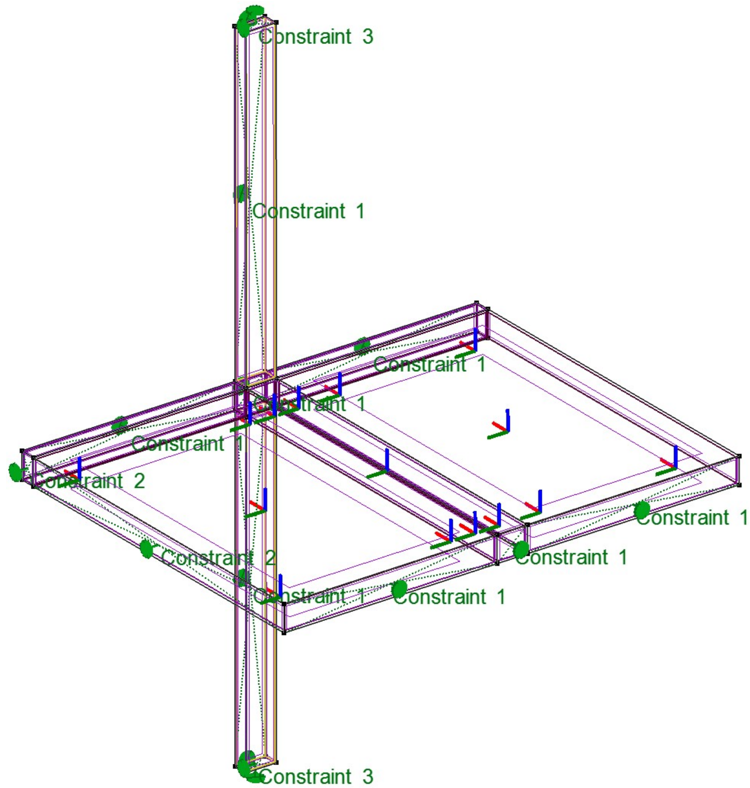


Figure 57: Boundary conditions in Interval 2

4.2.5 Task definition

This example contains 2 interval:

Table 53: List of intervals

Interval	Name	LC combination	Steps
1	Self weight	Supports, Weights, Load force	10
2	Creep after 10 years	Supports	100

5. TRANSPORT ANALYSIS

This chapter contains examples of transport analysis using the program **ATENA 2023 CeSTaR-2**.

5.1 FireStat Temp Effect On Bond

Example for modelling temperature related problems in ATENA. In this case the effect of temperature to reinforcement bond is investigated.

File name: fire_stat_temp_effect_on_bond.pre

5.1.1 Geometry

Geometry is presented in the following figures (Figure 58 and Figure 59)

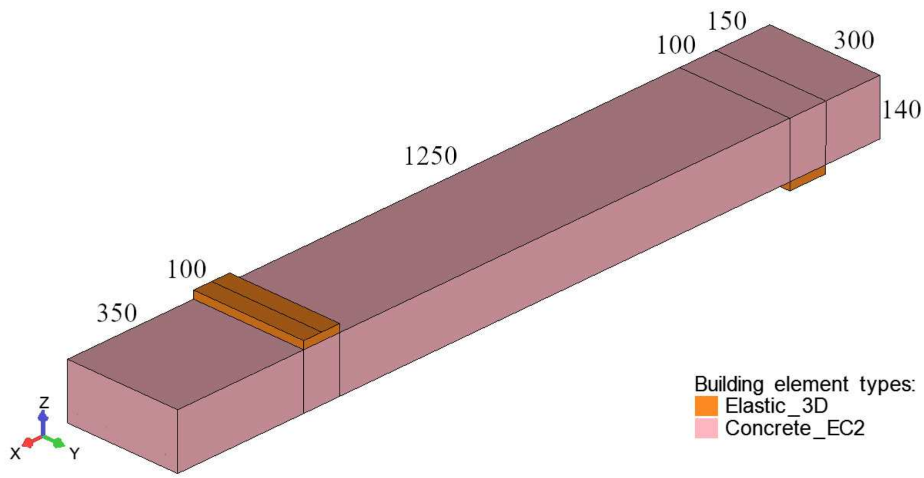


Figure 58: Solid model with boundary conditions. Dimensions in mm

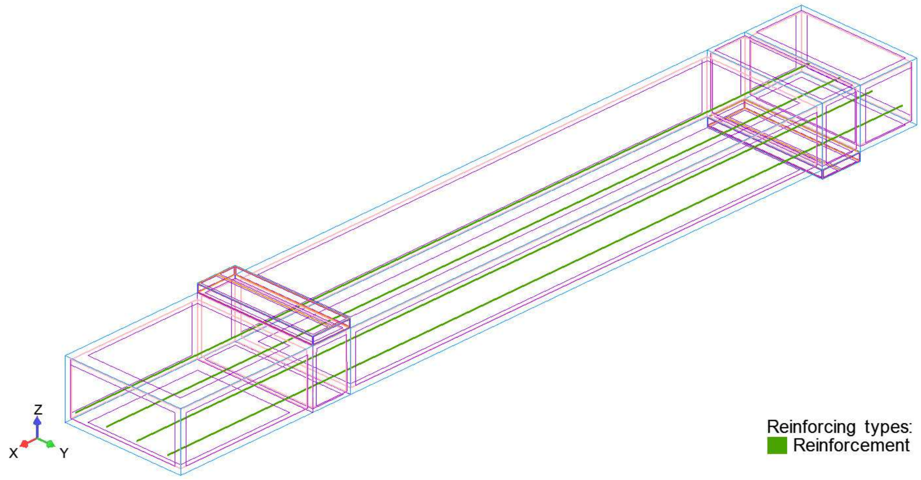


Figure 59: Wireframe model with reinforcement location

5.1.2 Materials

This section describes the materials used in this example.

Table 54: Used materials

Name	Prototype	Generator type	Base parameters	Assigned types
Elastic_3D	Elastic	Main	E=200000 MPa, μ=0.3, ρ=0.0025 kton/m³, α=1.2E-05 °C⁻¹	1, 18
Reinforcement_EC2	CCReinforcement	EC2	E=200000 MPa, ρ=0.00785 kton/m³, α=1.2E-05 °C⁻¹	18
Reinforcement	CCReinforcement	EC2	E=200000 MPa, ρ=0.00785 kton/m³, α=1.2E-05 °C⁻¹	13
Concrete_EC2	CC3DNonLinCementitious2	EuroCode2	E=34000 MPa, μ=0.2, ρ=0.0023 kton/m³, α=1.2E-05 °C⁻¹	18

5.1.3 Load cases

This example contains 3 load cases.

Table 55: Used load cases

Name	Load type	Load category	Multiplier	Boundary conditions
Supports	General	Undefined	1	1, 4, 5
Load force	General	Undefined	1	10

5.1.4 Boundary conditions

The following 2 boundary conditions were used in this example:

- Constraint
- Load force

Name	Condition type	Load case	Subject type	Geometry IDs
Constraint 1	Constraint	7 - Supports	Curve	66
Constraint 2	Constraint	7 - Supports	Surface	7
Constraint 3	Constraint	7 - Supports	Surface	9
LoadForce	LoadForce	8 - Load force	Point	9, 33

5.1.5 Task definition

This example contains 2 interval:

Table 56: List of intervals

Interval	Name	LC combination	Steps
1	Load force	Supports	30
2	Load heat	Supports	250
3	Load heat	Supports	10

5.2 Bridge Pier Hydratation 3D temp

A practical example of ATENA modelling of concrete hydration.

File name: bridge_pier_hydration_3Dtemp.pre

5.2.1 Geometry

Geometry is presented in the following figures (Figure 60 and Figure 61).

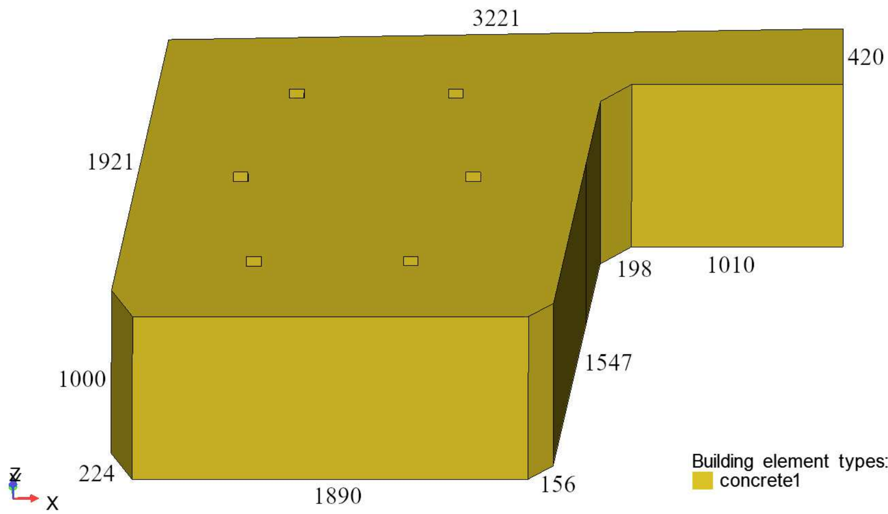


Figure 60: Solid model. Dimensions in mm

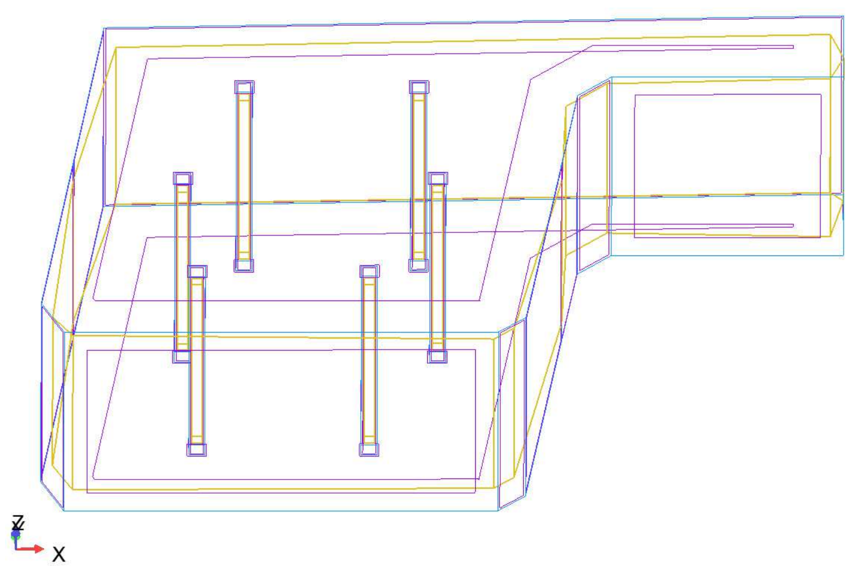


Figure 61: Wireframe model

5.2.2 Materials

This section describes the materials used in this example.

Table 57: Used materials

Name	Prototype	Generator type	Base parameters	Assigned types
concrete1	Transport	Main		2

5.2.3 Boundary conditions

The following 2 boundary conditions were used in this example:

- Dirichlet temperature
- Moisture temperature boundary

6. VALIDATION EXAMPLE

In this chapter, the three variants of cycling analysis of reinforced concrete is columns is presented. The columns were experimentally tested in Taiwan by the project partners. Numerical simulations in ATENA were performed on all three variants with various degrees of detail and simplification. Changes made are documented where necessary or where the model seemingly differs from the experimental setup. Notably, the figures referencing the experimental setup are borrowed from the Taiwanese side's Experimental Program Report they kindly provided during the solution of the project.

6.1 Common Characteristics for All Columns

6.1.1 Common Geometry

All columns are 2700 mm in height, have a bottom and top anchoring concrete block with dimensions of 1500×1500×1100 mm and 1100×1100×700 mm, respectively. These blocks were initially modelled as linear elastic macroelements for simplification, but simulation results showed that this is an oversimplification as damage is also occurring there. Therefore, energy dissipation occurs as well, and the blocks need to be considered in the non-linear analysis.

6.1.2 Common Loads

The simluations' loading protocols for all specimens consisted of these interval groups:

- Self-weight
- Pre-stressing and axial loads
- Cyclic displacement loading

Self-weight can only be represented by a single substep as there are no non-linear effects occurring and we can save on computation times. The same is true for pre-stressing and axial loads—we are well within the elastic range for our materials. Pre-stressing strands have strength of ca. 1600 MPa and we introduce only a few hundreds of MPa. Same holds true for the axial load as we only introduce a 10 % axial load $(F/A_g f_c')(F/A_g f_c')$, so concrete is in the elastic range as well. For the loading by displacement, we must consider that there is a great degree of non-linear behavior for both concrete and reinforcement. Therefore, division of each individual branch of the hysteresis curve into multiple substeps is necessary. From our experience with these analyses, the most practical and sufficiently accurate way to apply the load is to use a 6.75 mm line displacement condition (represents 0.25 % drift ratio for the column geometry—height is 2700 mm) for the displacer and specify interval multipliers and divisions that govern the amount of displacement and substep division in the current interval. An example of this procedure is shown on Figure 62. The loading histogram can be seen on Figure 63. To keep the computation times at manageable levels, it is recommended to only use one branch of displacement loading per drift magnitude (i.e. skipping drift magnitudes that occur in multiples—this includes negative values). It was found that this way, the accuracy of the simulation still very much resembles the full loading regime while reducing computation times by ca. 80 %. This reduction is recommended as we presume that if the histogram were kept in full, the simulation would have taken many days, considering a very recent computer with an 8-core, 16-thread 3.8 GHz processor and 64 GBs of 3200 MHz RAM.

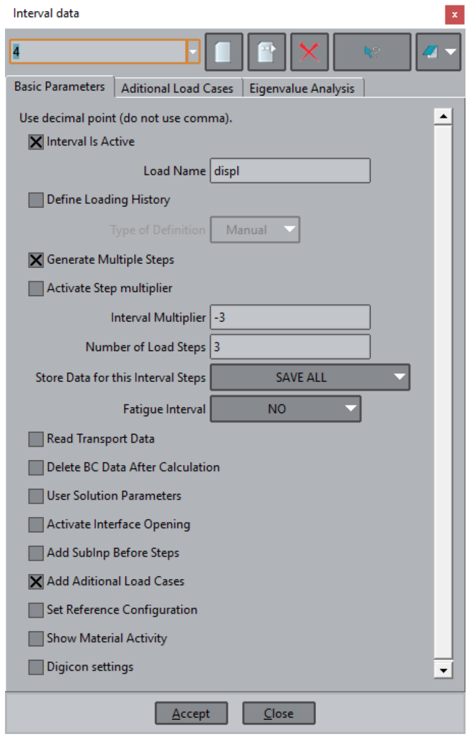


Figure 62: Interval setting—the second displacement interval. The 1st interval is self-weight, 2nd is pre-stress, 3rd is displacement up to 0.25% drift and this 4th interval represents -0.50% drift. The aforementioned recommendation is applied to skip steps with the same magnitude. The Interval Multiplier is negative if the load is to be applied in the opposite direction. Interval Multiplier specifies how many times a given load is applied and Number of Load Steps specifies the division into substeps. Here, we found that satisfactory results can be obtained if these two parameters are directly proportional.

Anchoring of specimens, which was experimentally achieved by using post-tensioned bolts into a strong floor, was modelled by using 1D elements representing the 69 mm diameter bars loaded by prestressing up to a value of 100 tf (ton-force) per bar (267 MPa for these lower, vertical bars). The upper bars, which are 36 mm in diameter, were prestressed up to 20 tf (192 MPa for the upper, horizontal bars). It is recommended to only use a one-step interval for this load as we are well within the elastic range for the given pre-stressing material and we can reduce the computation time.

The 100T MTS actuator, which delivered the loading by displacement, was modelled simply as a stiff displacer element meshed with tetrahedral elements. The displacer is not interesting in the analysis so its mesh can be rather coarse, and its material can be linear and elastic. As an example, see Figure 75, where the upper column detail is provided. To ensure that energy is not devoted to deformation of such an adjacent material that is only required to transfer loads, it is recommended to increase the value of Young's Modulus (ca. from conventional 200 or 210 GPa to 2000 GPa).

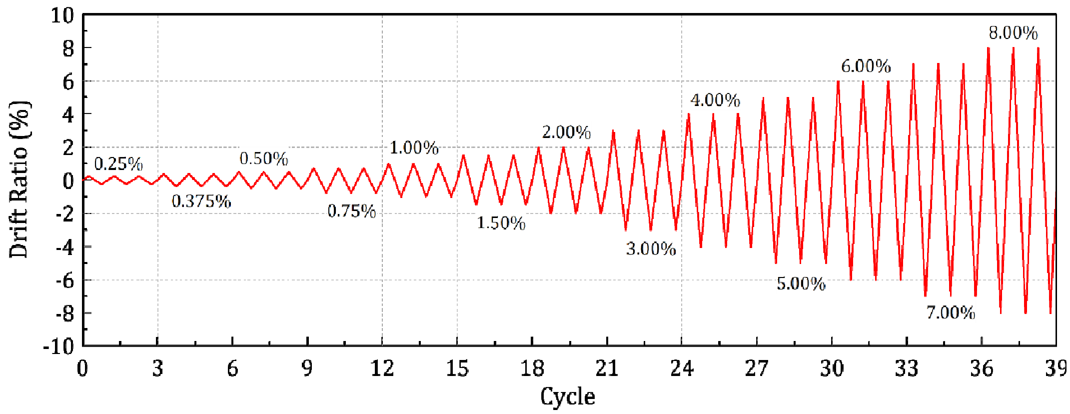


Figure 63: The prescribed loading by displacement. The columns are all 2700 mm tall so a drift ratio of 1 % represents a lateral load of 27 mm. That is a large step for the model so a substep with displacement of 0.25 % (6.75 mm) was chosen instead.

6.1.3 Mesh

The used mesh is the same for all column variants as they differ in configuration of reinforcement but not in the outer geometry. The model mainly uses linear hexahedral (brick) elements with emphasis on aspect ratio for a good quality mesh. Linear tetrahedral elements are used for the top displacer. Reinforcement is modelled as embedded, and its mesh is made as 1 element for each 1D member and subsequently division is made after the creation of the hexahedral elements for concrete. The lower column part, where damage is localized, has a finer mesh. That can be seen on Figure 64. From our experience, it is enough to describe the column width with about 5 elements, but the lower parts, where damage is localized, need refinement. Therefore, it is recommended to perform mesh refining, most conveniently made by division of the column microelement into two, specifying contact conditions and refining the mesh on the smaller, lower part. There, the width is described by 20 elements.

6.1.4 Simplifications

The main simplification that was used for columns CSC and CSCF is omitting the anchoring assembly for the self-centering pre-stressing cables. Although the cables are not pre-stressed, they are anchored in the top and bottom blocks and as the top moves laterally, force is generated in the unbonded cable that makes the cable act as a self-centering member. While in real conditions, the force needs to be properly distributed into the concrete to prevent slipping via a welded assembly with wedges displayed on Figure 65, it is redundant for FEM simulations. We initially modelled the assembly (Figure 66) but since we saw little to no benefit to it (computation time increases, it is not needed for reinforcing the blocks—there is little to no stress and the bars can be easily fixed by boundary conditions), we decided to omit it in the following analyses and recommend the user to do so as well.

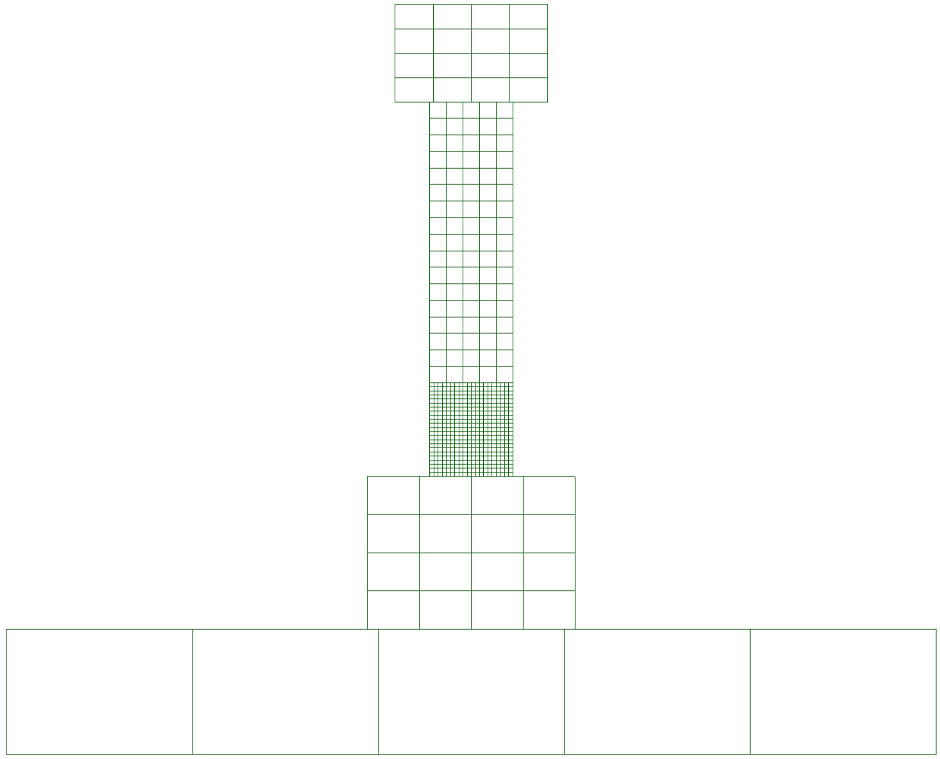


Figure 64: The bottom part of the column has a finer mesh than the rest. The greater part of the column is only described by 5 hexahedral elements, but the bottom part, where damage occurs, needs to be refined (20 elements). Note the stiff, elastic bottom ground needs only to be represented by a couple of elements to maintain aspect ratio. The top and bottom concrete anchoring blocks also do not need to have a very fine mesh (compared to the column).

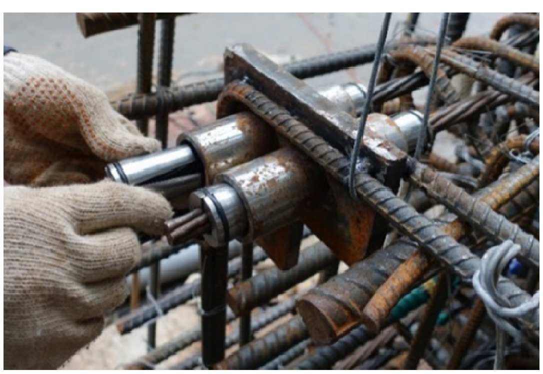


Figure 65: Wedges in the anchor during experiment.

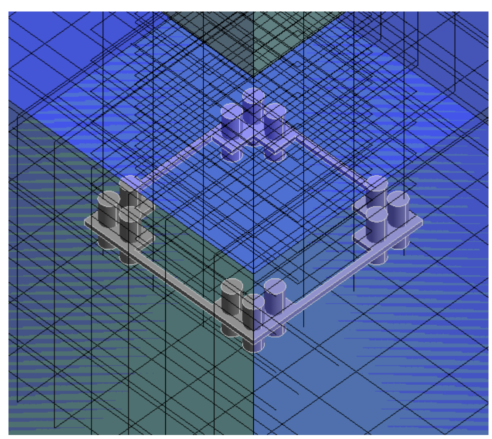


Figure 66: The abandoned FEA model of the anchoring members.

6.2 Material models

6.2.1 Concrete

The nonlinear behavior of concrete is examined using the ATENA program (Červenka et al., 2009) and the combined fracture-plastic model of Červenka & Pappanikolaou (2008). The material model NLCEM2 employed in this analysis assumes small strains and relies on a strain decomposition into elastic (ε_{ij}^{e}), plastic (ε_{ij}^{p}) and fracture (ε_{ij}^{f}) components. The model uses rate equations to describe the progressive degradation (concrete cracking) and plastic yielding (concrete crushing) of the material.

$$\dot{\sigma}_{ij} = D_{ijkl} \cdot (\dot{\varepsilon}_{kl} - \dot{\varepsilon}_{kl}^{p} - \dot{\varepsilon}_{kl}^{f}) \qquad (1)$$

The flow rule for the model governs the evolution of plastic and fracturing strains:

Plastic Model: $\dot{\varepsilon}_{ij}^{p} = \dot{\lambda}^{p} \cdot m_{ij}^{p}, \quad m_{ij}^{p} = \dfrac{\partial g^{p}}{\partial \sigma_{ij}}$ (2)

Fracture Model: $\dot{\varepsilon}_{ij}^{f} = \dot{\lambda}^{f} \cdot m_{ij}^{f}, \quad m_{ij}^{f} = \dfrac{\partial g^{f}}{\partial \sigma_{ij}}$ (3)

Where $\dot{\lambda}^{p}$ is the plastic multiplier rate, $\dot{\lambda}^{f}$ is the inelastic fracturing multiplier and g^{p} and g^{f} are plastic potential function and the potential defining the direction of inelastic fracturing strains, respectively. The model uses the consistency conditions to evaluate the change of the plastic and fracturing multipliers.

$$\dot{f}^{p} = n_{ij}^{p} \cdot \dot{\sigma}_{ij} + H^{p} \cdot \dot{\lambda}^{p} = 0, \quad n_{ij}^{p} = \frac{\partial f^{p}}{\partial \sigma_{ij}} \qquad (4)$$

$$\dot{f}^{f} = n_{ij}^{f} \cdot \dot{\sigma}_{ij} + H^{f} \cdot \dot{\lambda}^{f} = 0, \quad n_{ij}^{f} = \frac{\partial f^{f}}{\partial \sigma_{ij}} \qquad (5)$$

The model uses the Rankine criterion for tensile fracture with exponential softening of Hordijk (1991) and Menentrey & Willam (1995) to model the compressive behavior of the concrete. Hardening and softening are defined according to the laws described in Figure 69, where $\varepsilon_{eq}{}^{p}$ is the equivalent plastic strain. The crack band size and the crush band size are adjusted based on the crack orientation approach proposed by Červenka V. et al. (1995) which reflects the fact that a crack cannot localize into a single element if the crack direction is not aligned with the element edges.

$L_t' = \gamma L_t$ and $L_c' = \gamma L_c$

$$\gamma = 1 + (\gamma_{max} - 1)\frac{\theta}{45}, \quad \theta \in \langle 0; 45 \rangle \tag{6}$$

In nonlinear analysis of reinforced concrete, it becomes important to consider additional special issues related to the reinforcement and the composite reinforced concrete material. Some of the most important phenomena are:

- Shear strength and stiffness of cracked concrete
- Compressive strength reduction due to crack opening in perpendicular direction
- Reinforcement yielding
- Tension stiffening
- Dowel action and bending stiffness of the reinforcement
- Bond failure between concrete and reinforcement

The modified compression field theory of Collins (Bentz et al. 2006) is used to consider the first and second items. In this theory, the compressive strength is reduced by a formula that considers the tensile strain in the crack.

$$\sigma_c = r_c f_c' \tag{7}$$

$$r_c = \frac{1}{0.8 + 170\varepsilon_1}, \; r_c^{\lim} \le r_c \le 1.0 \tag{8}$$

Where ε_1 is the tensile strain in the crack. In ATENA the largest maximal fracturing strain is used for ε_1 and the compressive strength reduction is limited by $r_c^{\lim}$. In this work $r_c^{\lim}$ = 0.6 m.

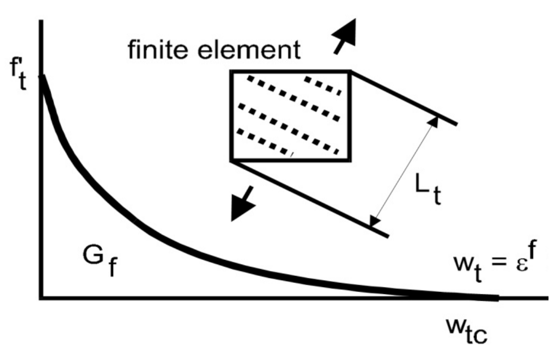


Figure 67: Tensile softening (Hordijk 1991).

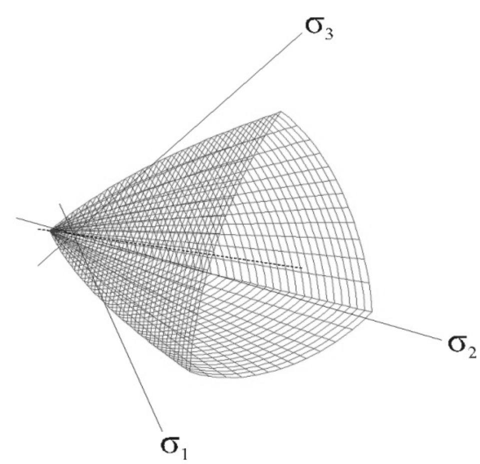


Figure 68: Concrete failure criterion (Menetrey & Willam 1995).

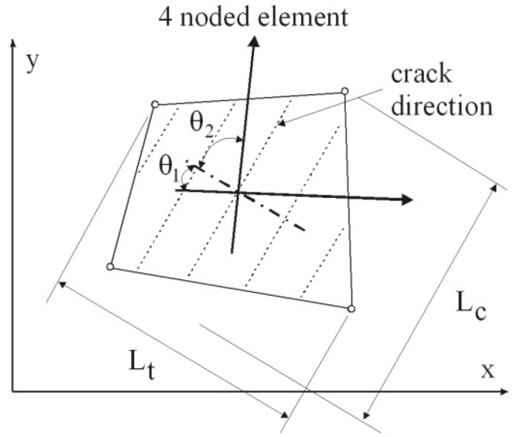


Figure 69: Crack band size adjustment based on crack direction orientation.

The shear strength of cracked concrete is computed using the modified compression field theory (MCFT) as outlined by Bentz et al. in 2006. The formula for this calculation is shown in Eq. 9, where f'_c is the compressive strength in MPa, a_g is the maximum aggregate size in mm, and w is the maximum crack width in mm at the given location.

$$\sigma_{ij} \leq \frac{0.18\sqrt{f_c'}}{0.31+\frac{24w}{a_g+16}}; \quad i \neq j \tag{9}$$

Although MCFT provides a formula for the shear strength, it does not give any information on shear stiffness, which is a critical parameter that greatly affects the performance of reinforced concrete. In our simulations, shear stiffness, represented by K_t^{cr}, is calculated using a scaling factor s_F, which is directly derived from the crack normal stiffness as shown in Eq. 10. This approach makes the shear stiffness dependent on the crack opening displacement.

$$K_t^{cr} = s_F\, K_n^{cr} \tag{10}$$

$$K_n^{cr} = f_t(w_t) / w_t \tag{11}$$

The NLCEM2 material model parameters required for these simulations were determined with expertise from the previous project between the partners. Here, they are listed:

- Young's Modulus—unfortunately, no experiments of this property were performed, so the user has to specify the modulus in the next best way, which is according to the code they select (EC2, ModelCode, SP63) based on the concrete's strength. The best way to determine the modulus is to perform cubic or cylinder axial tests and evaluate the data based on the stress-strain curve.
- Poisson's ratio—valid range is <-1 ; 0.5>. It can be specified via experiment, but 0.2 is usually a good value for concrete.

- Tensile Strength FT—valid range is <0 ; -FC/2>. Can be either generated from a code formula or specified directlyif the user has experimental data available.
- Compression Strength FC—has to be specified as a negative value (compression). Ideal input is from stress-strain diagrams of concrete cube or cylinder tests. In this case, this data was available.
- Fracture Energy GF—this parameter is not easy to obtain as conventional tests do not account for it. The best input is data from a specialized experiment. In our case, we specified a conventional value for the chosen concrete GF=1.5E-4 MN/m.
- Plastic Strain EPS CP—valid range <min ; 0>, generation formula is FC/E. Again, best input is in the form of a specialized test where peak stress occurs. Also influenced by quality of concrete. For this module, EPS CP= -1.3E-3.
- Onset of Crushing FC0—this parameter is also obtainable from experiment. Its generation forula is -FT*2.1.
- Critical Compressive Displacement WD—from our experience, this kind of cyclical loading is usually best described by WD = -0.0005 m. It is subject to change based on the expected ductility of the column.
- Fc reduction—describes the reduction in compressive strength because of cracking. The default value is 0.5. In our experience, it should be specified as 1.0 for cyclic loading as it is expected that spalling eliminates the concrete completely.
- Direction of plastic flow BETA—BETA influences the plastic flow of the structure in the post-peak region. The valid range for this parameter is <-5 ; 5>, default value is 0. For cyclic analyses, the optimal value is BETA = 0.5.

For more information oneach of these material parameters, please see the ATENA manual (Červenka 2009, also found in the directory of installation).

6.2.2 Reinforcement and Bond

To model reinforcement, the embedded approach with truss elements is used, and a multi-linear stress-strain law is employed to account for reinforcement yielding. The model also has the option to include tension stiffening.

Reinforcement is modelled with a multilinear stress-strain curve that can use experimental stress-strain data as input. It uses the CCCyclicReinforcement model. The reinforcing bars are modelled as bars with memory bond for the cyclic analyses. This analysis uses the 2010 CEB FIB model code (Walraven 2010) to model reinforcement bond with concrete. Individual bond parameters are generated based on the mean compressive strength of concrete and rebar parameters. More on bond in Červenka 2009.

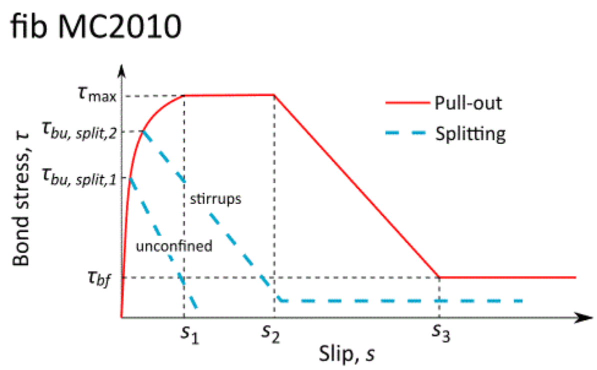


Figure 70: The CEB FIB 2010 model code bond expression used in this ATENA module.

6.2.3 Solid Elastic

The solid elastic material model is very simple as it is basically only used for macroelements that transfer loads into other parts of the structure. Being linear, the model excludes any damage and we are, therefore, only left with specifying Poisson's ratio and Young's Modulus:

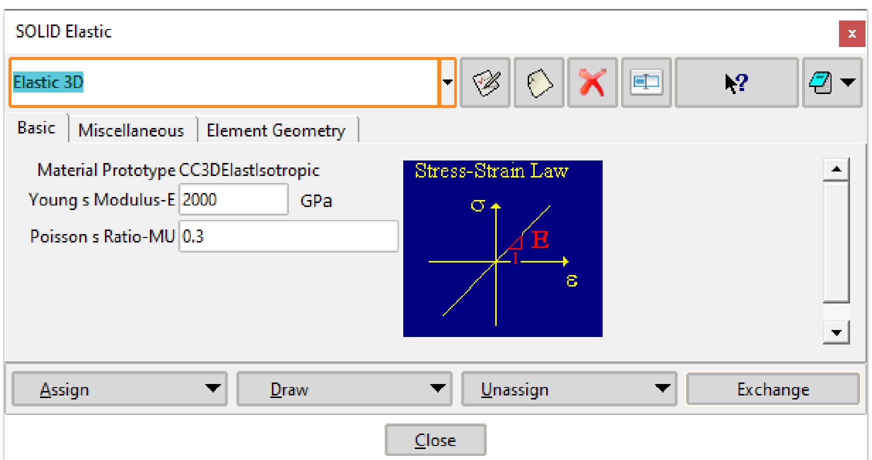


Figure 71: Material dialogue window for the solid elastic material used to model the displacer and foundation.

6.3 Modelling of Individual Column Variants

The basic tutorial on how to use this module will be described on the first column (CCC) for all column variants. The following variants will then have their own space to describe additional processes when the occasion arises. The process of modelling the self-centering un-prestressed cables will be described on the CSC column.

6.3.1 CCC Variant

This first, basic column variant only has rectilinear stirrups (no spiral reinforcement) and no self-centering cables like the other variants. It is important to accurately describe the placement of the reinforcement (at least because it differs from variant to variant) as reinforcement configuration and overall percentage of reinforcement in the concrete have great effect on the response. Notably, the stirrups do not need to be represented by circular elements and with overlapping hooks, but rather by linear elements with boundary conditions that specify fixed starts/ends. The same is true for the longitudinal reinforcement—no need to model hooks if present—just apply boundary conditions (fixed start/end). This is illustrated on Figure 79.

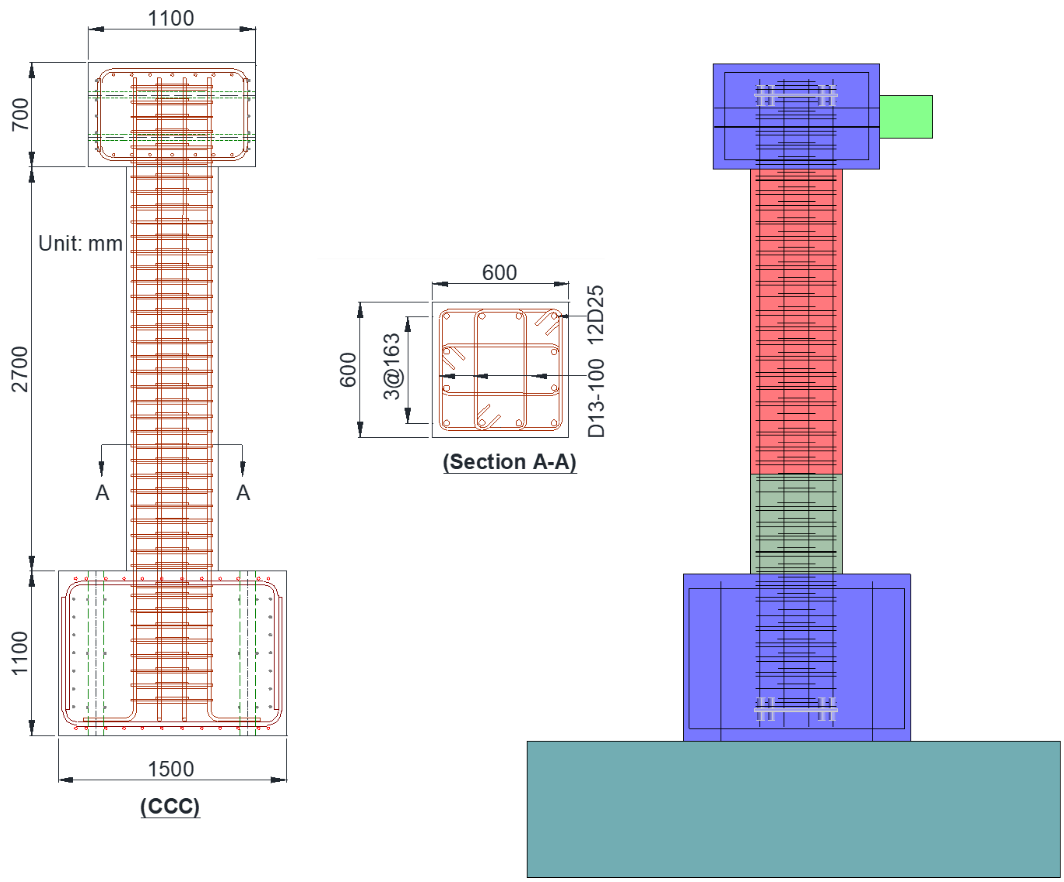


Figure 72: The CCC column setup documentation (left) and its GiD model (right). Inside the top and bottom anchoring blocks, there is outlined a frozen layer of the anchors described on Figure 65. These are not a part of the mesh and have not been modelled in the final analyses.

Figure 73: The anchoring parts of longitudinal reinforcement do not need to be modelled in any variant. Rather, apply fixed start/end condition from the material window (Figure 79).

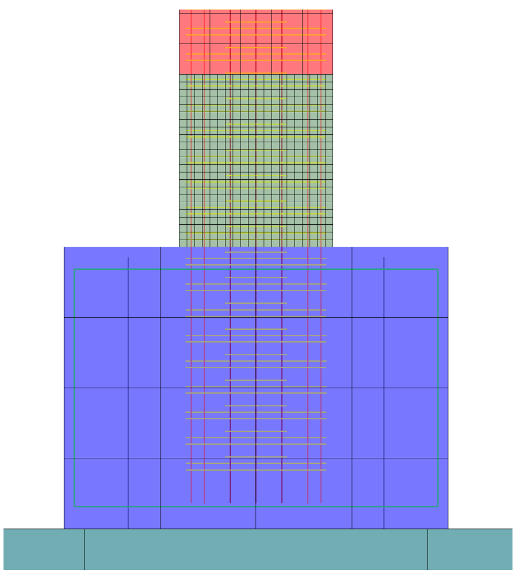


Figure 74: Lower CCC column detail. The bottom block is anchored into the ground by 4 big metal rods.

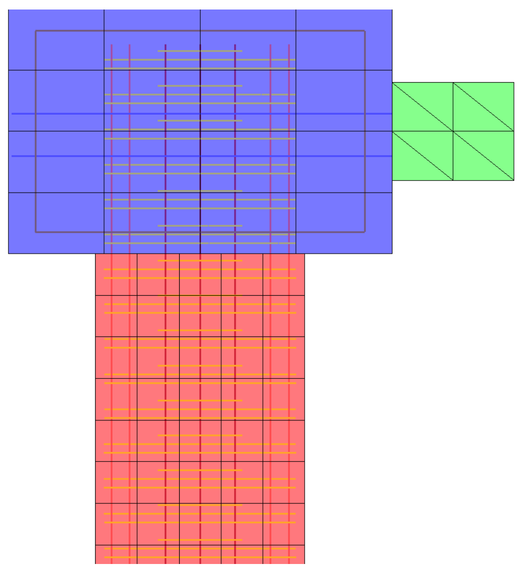


Figure 75: Upper CCC column detail. The green color describes the displacer used to apply the load.

The qualitative parameter chosen for the experiments was the force-drift diagram that uses the measured values of lateral reaction force plotted against the column drift that is calculated from 2 averaged displacement measurements at the top of the column's height. This way, the user has a parameter that can be used as a benchmark to evaluate other potential column variants. To create this diagram, we need to apply the displacement load and set up monitors for the 1st interval. This is illustrated on Figure 76 as a red arrow. Then, to plot the Y axis, we must select the feature "MaxMonitor for Surface" to sum up all the reactions on the *x* face of the displacer and then we can plot it against the displacement, or drift, if we choose to do so. The results of the CCC variant together with the experimental data are shown on **Chyba! Nenalezen zdroj odkazů.**.

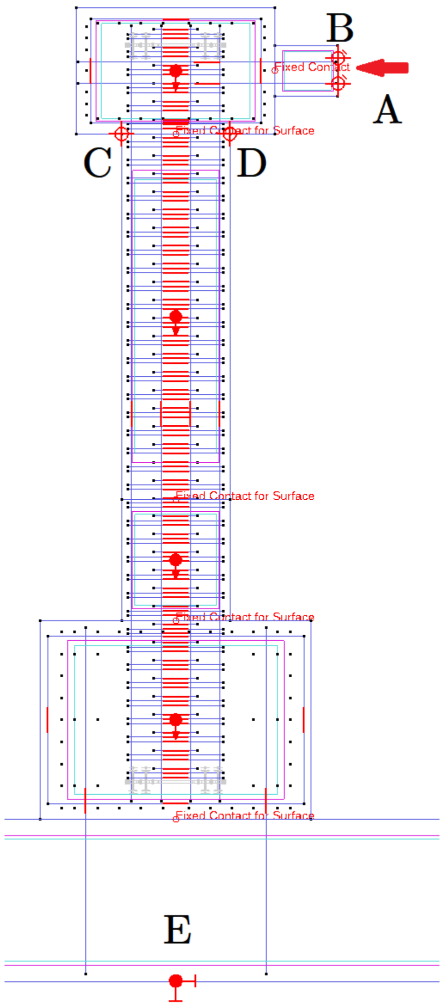


Figure 76: Boundary conditions of the CCC column. A) displacement loading, B) reactions monitor, C) x+ displacement monitor, D) x- displacement monitor, E) Fixed boundary condition.

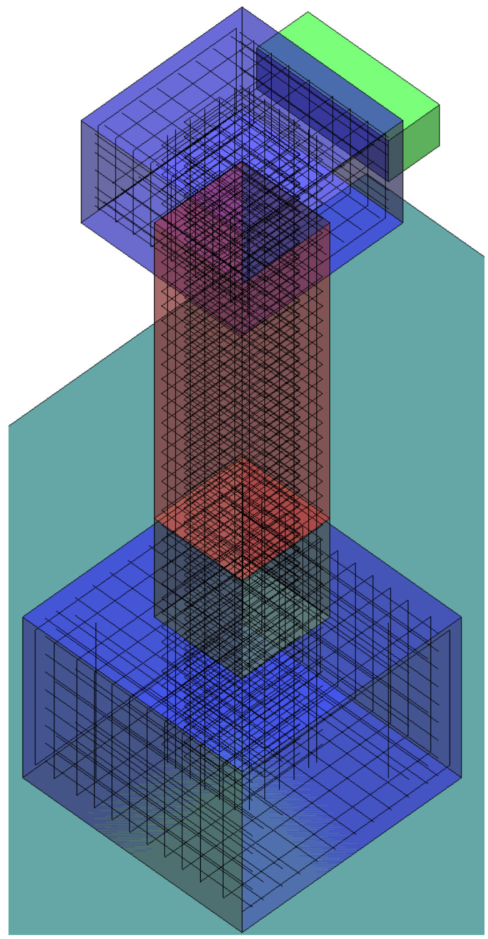


Figure 77: Axisymmetric view of the CCC column model with reinforcement.

The second column variant has a little different longitudinal reinforcement configuration, and we must account for that. More importantly, it has a unique self-centering un-prestressed strand configuration that acts as a stabilizing member during lateral load. The main idea of this reinforcement is to eliminate bond in the middle parts of the column to allow force to be generated inside the strands that acts as a countermeasure during displacement loading. The cables are shown on Figure 78.

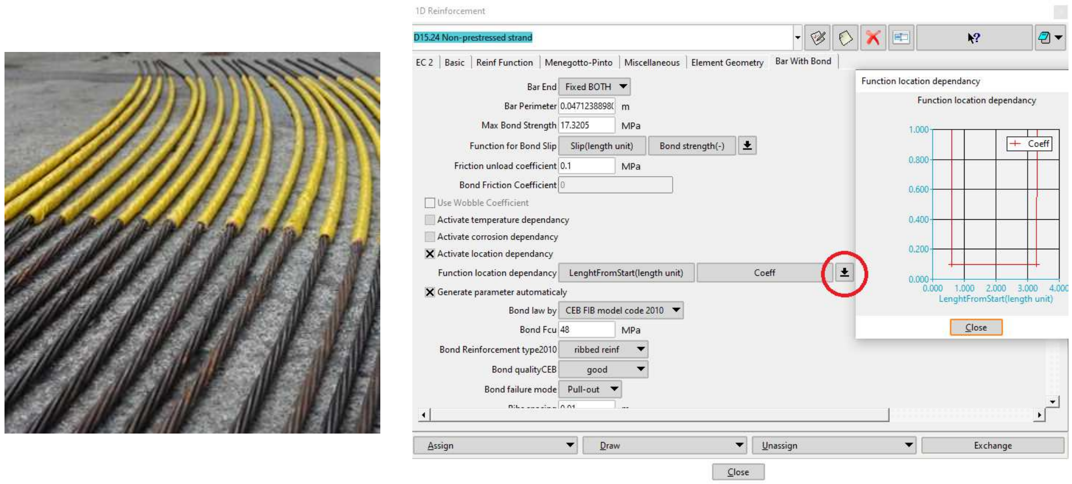


Figure 78: Pre-stressing cables used to generate a stabilizing force in the columns. In the experiment, a yellow PVC material was used to separate the reinforcement from concrete to eliminate bond. In the analyses, this was initially done by separating a 1D reinforcement element into 3 parts to simulate the bond conditions. Subsequently, ATENA modules for GiD were updated to implement this feature so the user now only has to model 1 element and can specify bond length across the bar for ease of operation. The red highlighted button now opens up a tabular data window where the lengths can be specified. Then, we can plot a bond-bar length graph shown on the right-hand side.

The documentation and the GiD model for this variant is shown on Figure 80. The anchoring members with bearing plates and wedges were not modelled. Rather, a quick boundary condition can be specified instead to save computation times if the user feels like the contribution of the plate to the bearing capacity is negligible (here it is as damage is localized in the column).

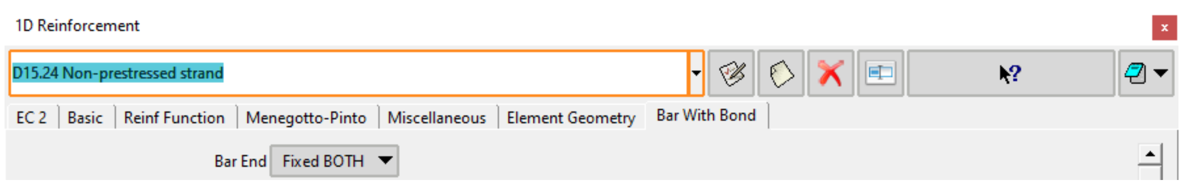


Figure 79: Dialogue window of the 1D reinforcement element for the fixed, non-prestressed strand.

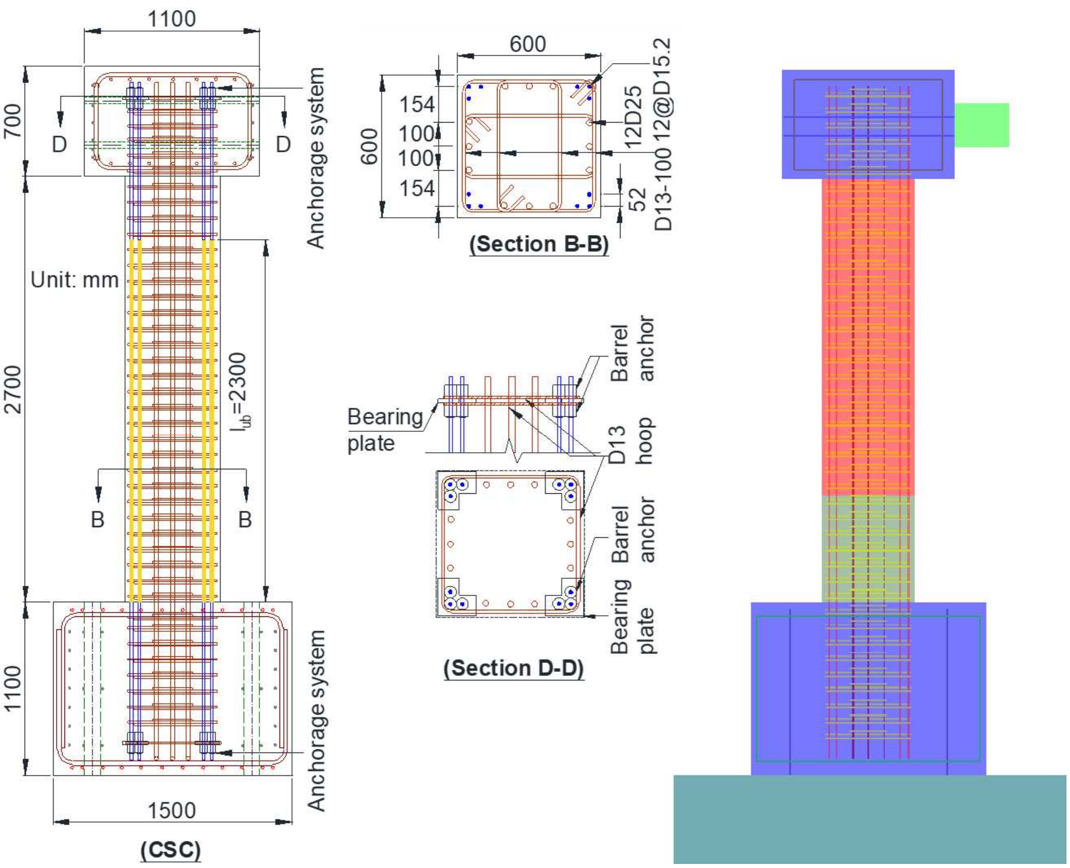


Figure 80: The CSC column setup documentation (left) and its GiD model (right). The yellow parts correspond with Figure 78, where they represent the length of suppressed bond by using PVC.

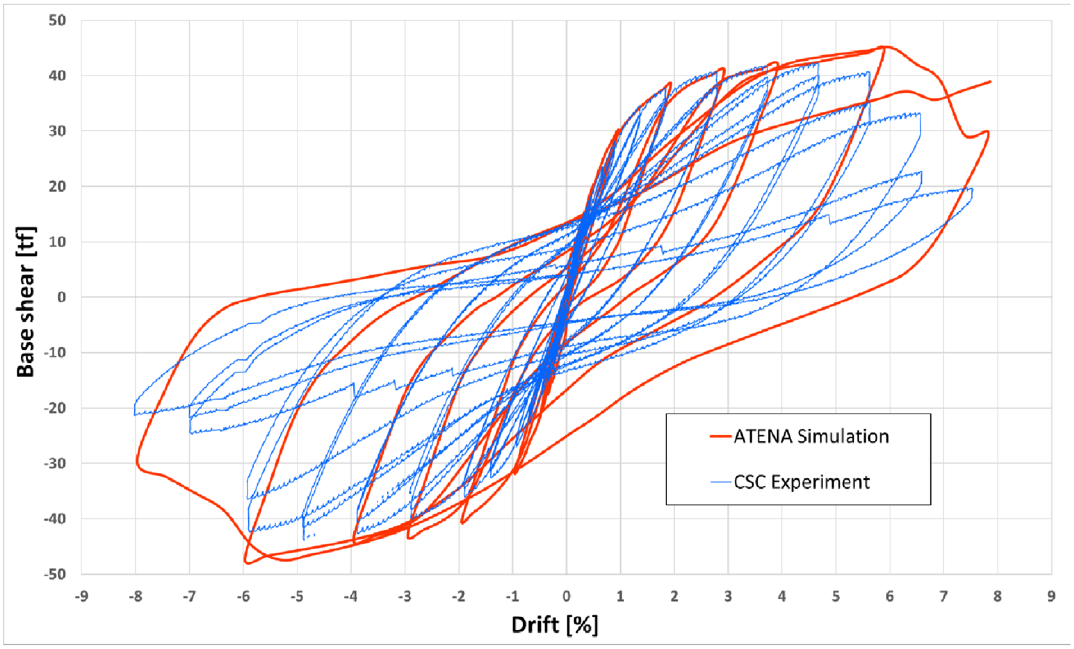


Figure 81: The CSC variant has a higher response compared to the CCC variant—mainly due to the presence of the un-prestressed strands. The model has a tendency to continue with more hardening while the experiment shows the hardening is not as prevalent. However, there is still very good agreement between the ATENA-CeSTaR simulation and the experiment.

6.3.2 CSCF Variant

The CSCF variant presents a combination of two novel approaches—one being the self-centering un-prestressed strands and the other being the multispiral reinforcement. For modelling of circular elements, spirals, helices etc., the ATENA GiD module provides the user with curved members so that modelling by discretization into lines is not necessary.

The development of modelling and simulation instruments also included methods and algorithms for effective definition and parametrization of pre-cast structural members for numerical modelling. Notably, the following:

- Development of methods and algorithms for definition of the numerical model on a CAD and BIM basis with support of curved entities on the basis of NURBS curves and surfaces.
- Development of methods for parametrization of structural geometry and multi-spiral reinforcement

The user can now use NURBS curves for definition of multi-spiral reinforcement with the aid of BIM data (Figure 82). Figure 83 shows use of the Python scripting language for parametrization of the task. This way, the user can parametrize the dimensions, pitch and radius of the multi-spiral reinforcement.

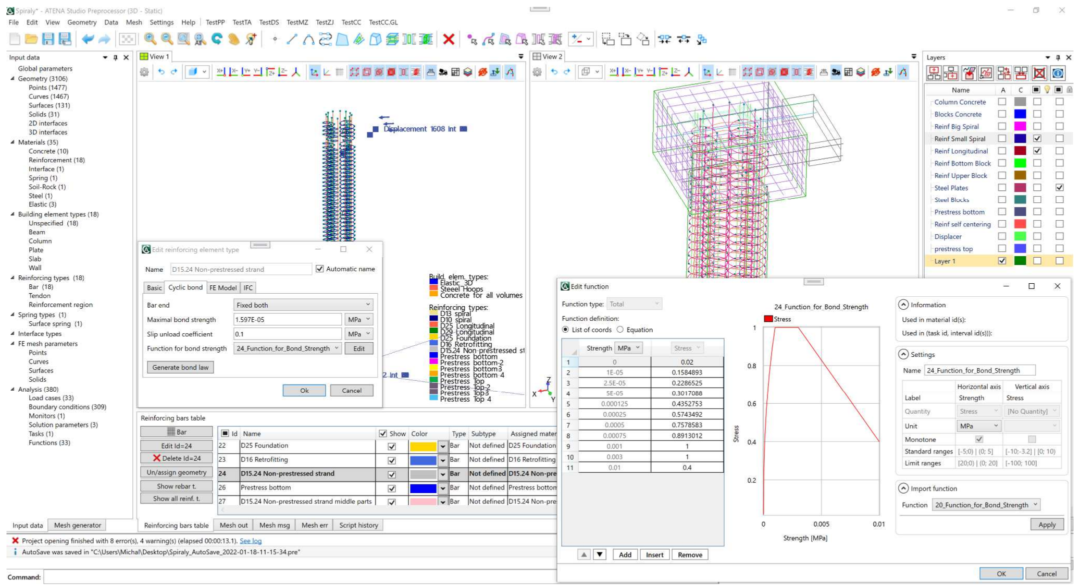


Figure 82: Structure with a spiral reinforcement in the ATENA-CeSTaR software.

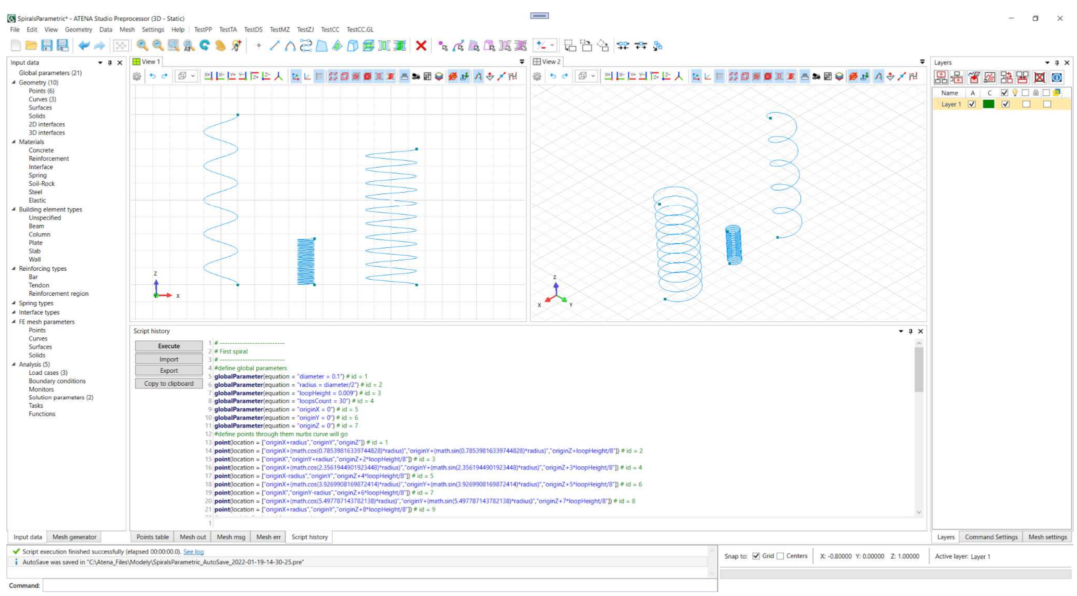
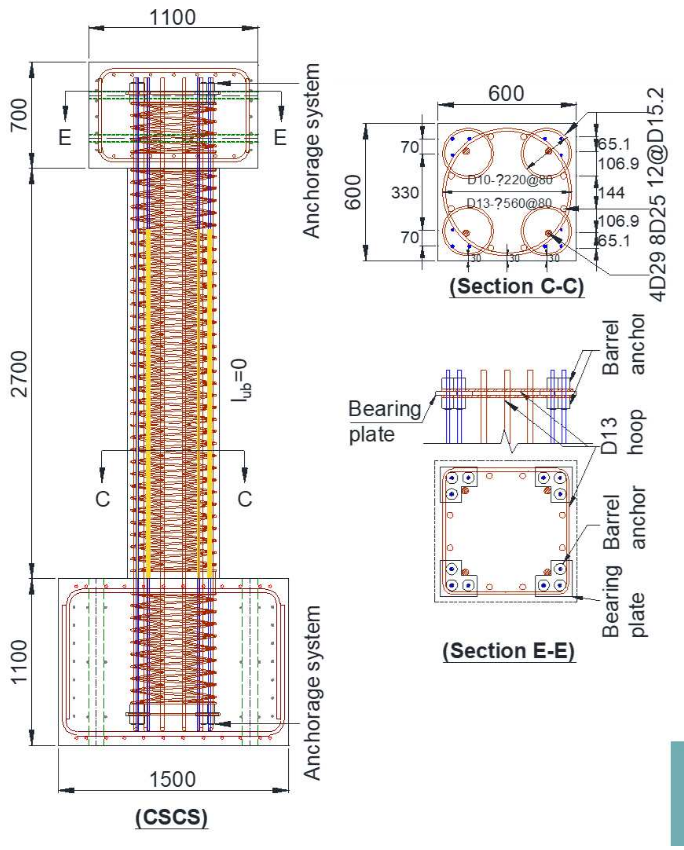


Figure 83 Parametrization of the input using Python scripts in the ATENA-CeSTaR software.



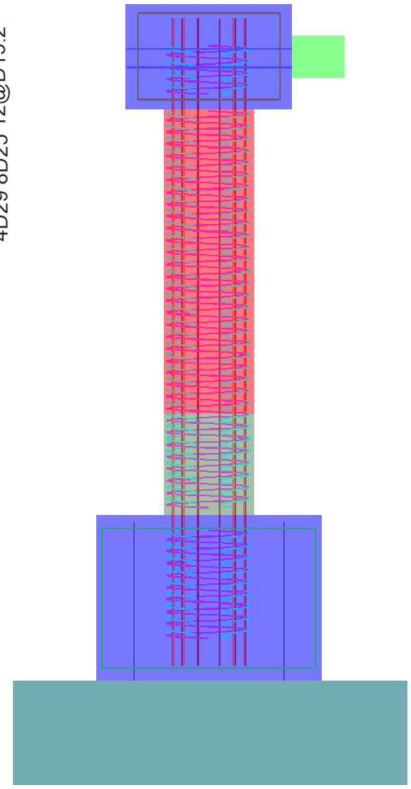


Figure 84: The CSCF column variant documentation and GiD model. The column has different longitudinal reinforcement, different strand positions and uses spirals instead of stirrups. Again, the anchoring plate was not modelled.

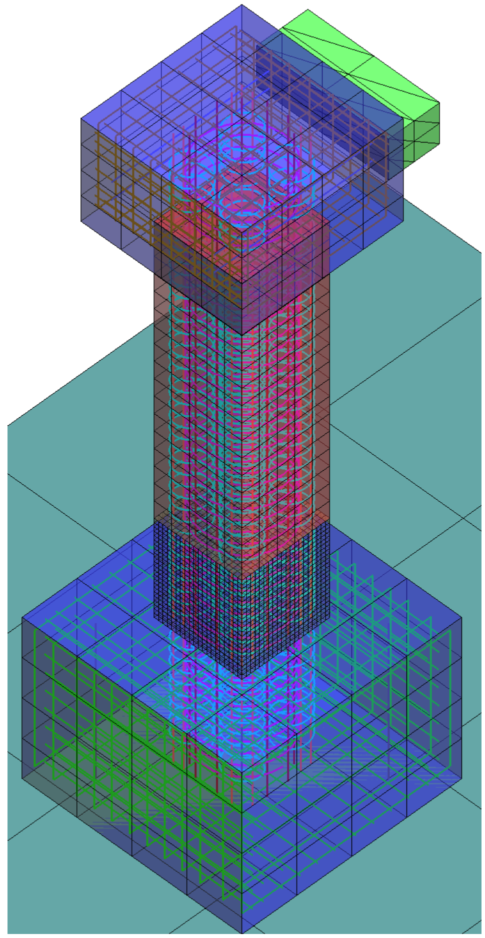


Figure 85: Detailed view of the reinforcement modelled in the CSCF variant.

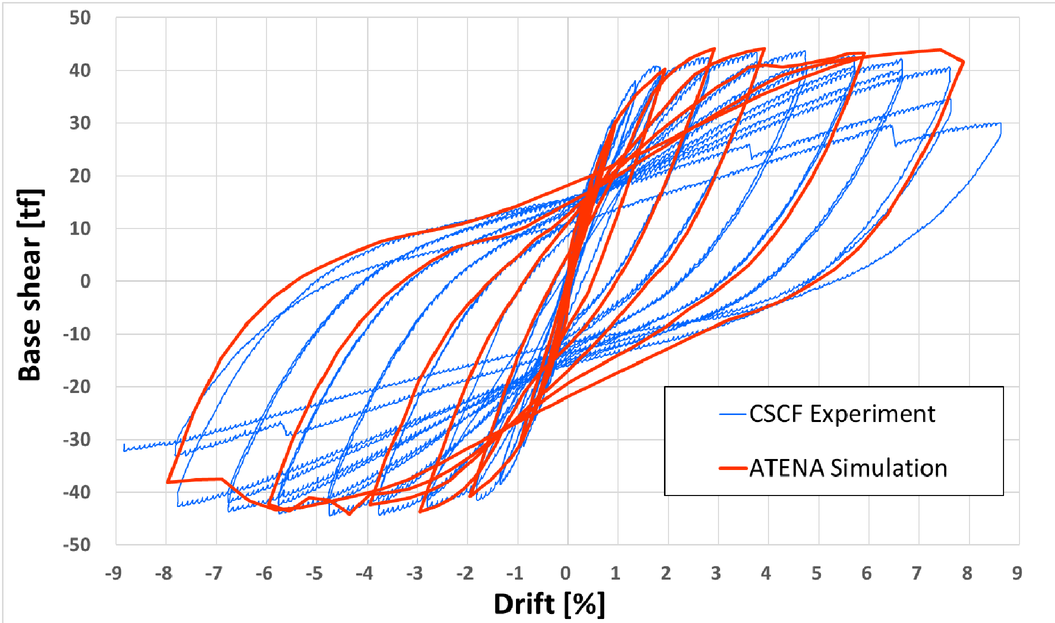


Figure 86 The CSCF column has the potential to have the best response to the cyclic lateral loading as the spiral reinforcement adds additional stiffness in the x-y direction and also provides confinement—therefore, this column variant has the highest response both in experiment and in simulations (mainly in post-peak areas, where it maintains good base shear compared to the CCC variant). The CSCF variant has the best agreement with the experiment among all variants.

7. SUMMARY

This document summarizes the example data and validation problems for the new CeSTaR-2 module in ATENA 2023 software.

The first part of the document describes the example problems that are part of the software installation. They include examples of reinforced concrete analysis in Statics, Dynamics, Creep and Transport.

Chapter 6 of this document describes in more detail input data definition, material parameters, solution strategy for problems that were experimentally tested during the project by the Taiwaneese partners. It also represents the Validation of the developer software module. The three column variants (CCC, CSC and CSCF) with rectilinear and multispiral reinforcement and the CSC and CSCF variants also with the self-centering strands under cyclic loading were analyzed and evaluated.

The chapter explains the process of creating the geometric model, common loads, simplifications, mesh and the user interface as well as important details about boundary conditions, solution parameters, and material models. The simulations in the ATENA module were conducted using experimental data and parameters outlined in the manual's chapters. The manual also includes graphical outputs and explanations on how to evaluate the physical properties of concrete and reinforcement using the ATENA Studio Postprocessor.

8. REFERENCES

[1] Larson, A. (2011). Sustainability, Innovation and Entrepreneurship. Flat World Publishing.

[2] Bureau of Energy, Ministry of Economic Affairs of Taiwan. (2019). Monthly Accumulated Data of Energy Statistics.

[3] Kuo, M.T. (2008). Axial Compression Tests and Optimization Study of 5-Spiral Rectangular RC Columns. Master Thesis, National Chiao Tung University.

[4] Hsu, W.C. (2019). Cyclic Behavior of Concrete Columns with Unstressed Steel Strands as Longitudinal Reinforcement. Master Thesis, National Taiwan University.

[5] Červenka, J., Jendele, L., Červenka, V. 2009. ATENA Program documentation. Cervenka Consulting, www.cervenka.cz.

[6] Červenka, J., Pappanikolaou, V. “Three dimensional combined fracture-plastic material model for concrete”. Int. J. of Plasticity, Vol. 24, 12, 2008, ISSN 0749-6419, pp. 2192-2220.

[7] Hordijk, P. (1991). Local Approach to Fatigue of Concrete. Doctoral Thesis, Delft University.

[8] Menetrey, P. and K. J. Willam (1995). "Triaxial Failure Criterion for Concrete and its Generalization." ACI Structural Journal 92(3): 311-318.

[9] Cervenka, V., R. Pukl, et al. (1995). Mesh Sensitivity Effects in Smeared Finite Element Analysis of Concrete Fracture. FRAMCOS2, Zurich, Aedificatio.

[10] Bentz, E. C., Vecchio, F. J., & Collins, M. P. (2006). Simplified modified compression field theory for calculating shear strength of reinforced concrete elements. ACI structural journal, 103(4), 614.Walraven, J. C. (2010). The 2010 fib model code for concrete structures. Concrete Plant International, 2010(5), 32-37.