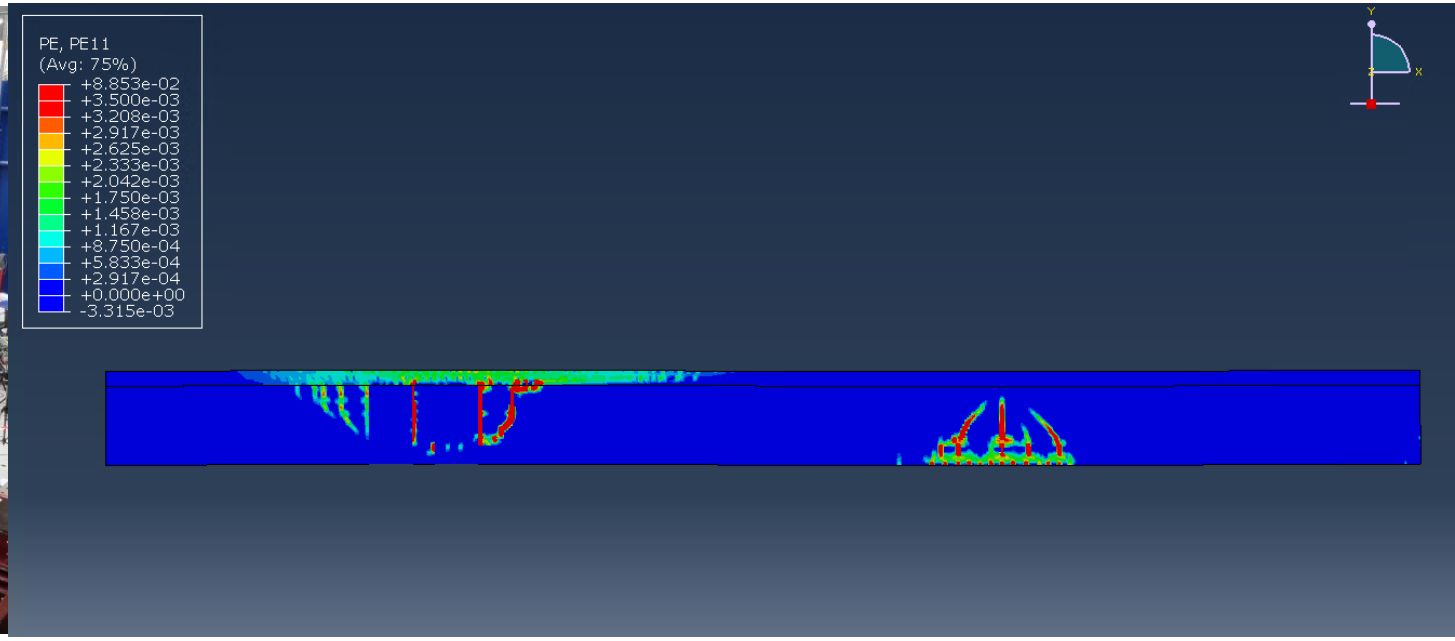
www.abt.eu

Beam prediction contest 2023, TU Delft

Krishna Ajithkumar Pillai



TU Delft Delft University of Technology

fib CEB-FIP

Rijkswaterstaat
Ministry of Infrastructure and Water Management

Introduction

Krishna Ajithkumar Pillai
Junior FEA Specialist, ABT

k.pillai@abt.eu

DIANA FEA

- ✓ Advanced material models for concrete and bond slip reinforcement
- ✓ Acceptance by authorities such as RWS
- ✓ Lots of user experience

- – 3D meshing capabilities limited
- – No explicit solver

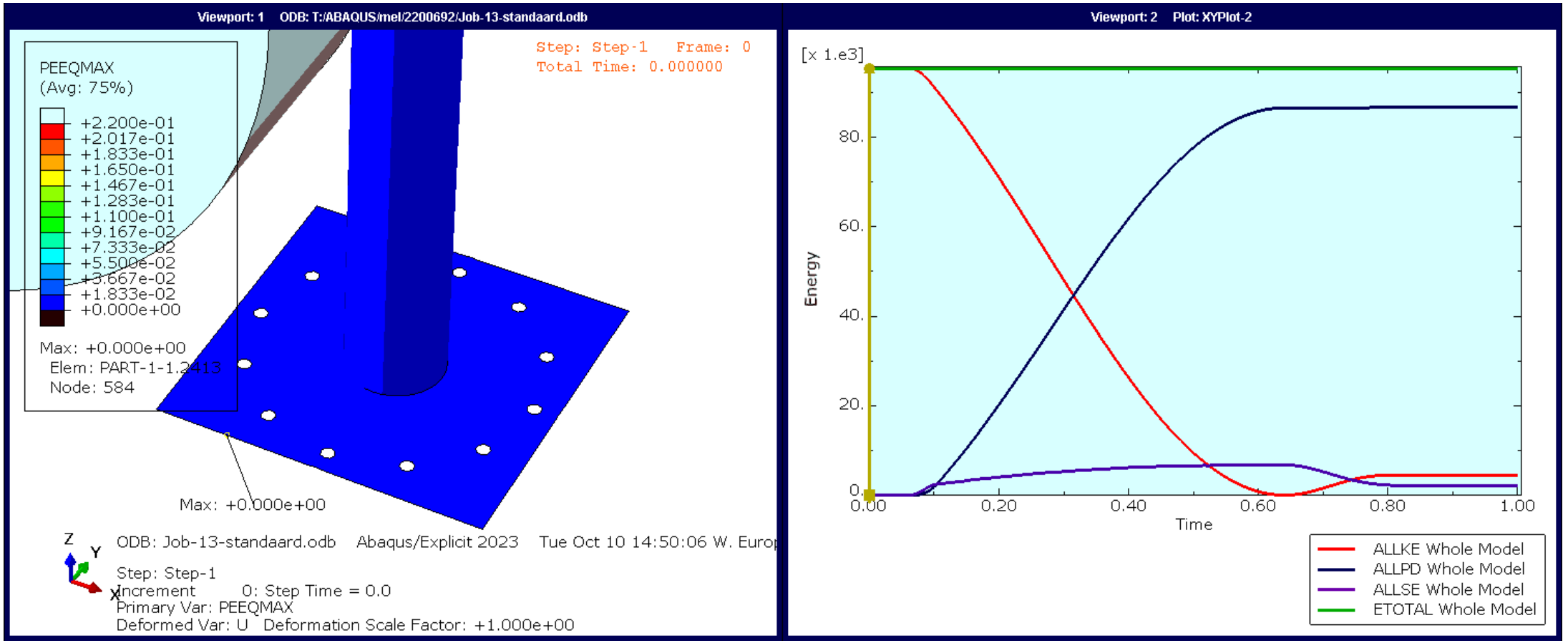
SIMULIA ABAQUS

- ✓ Advanced 3D meshing capabilities
- ✓ Support for parallel processing (CPU, GPU)
- ✓ Explicit solver
- ✓ Advanced contact models

- – Not used often in the construction industry
- – Less specifically aimed at structural / civil engineering

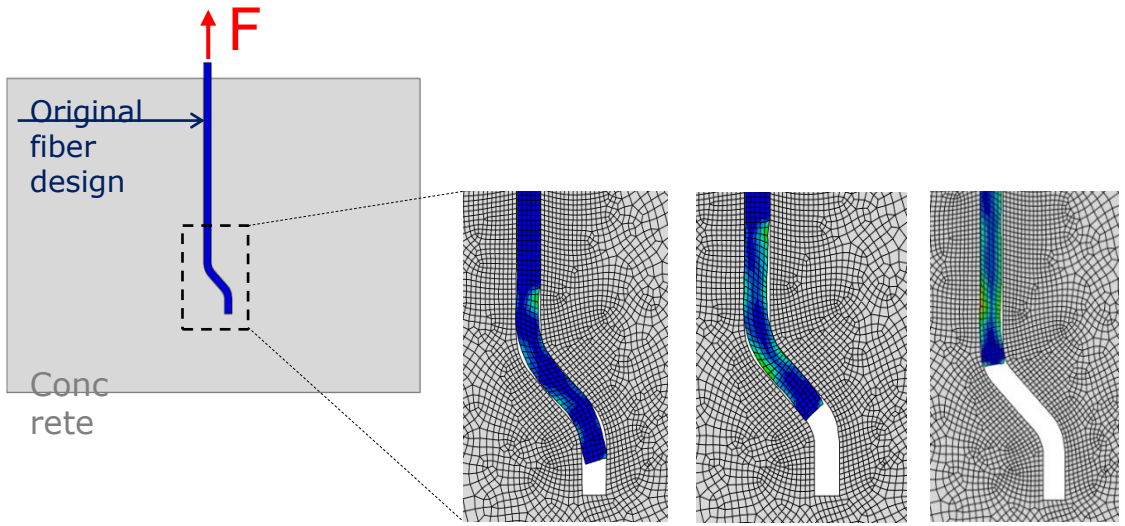
ABAQUS at ABT

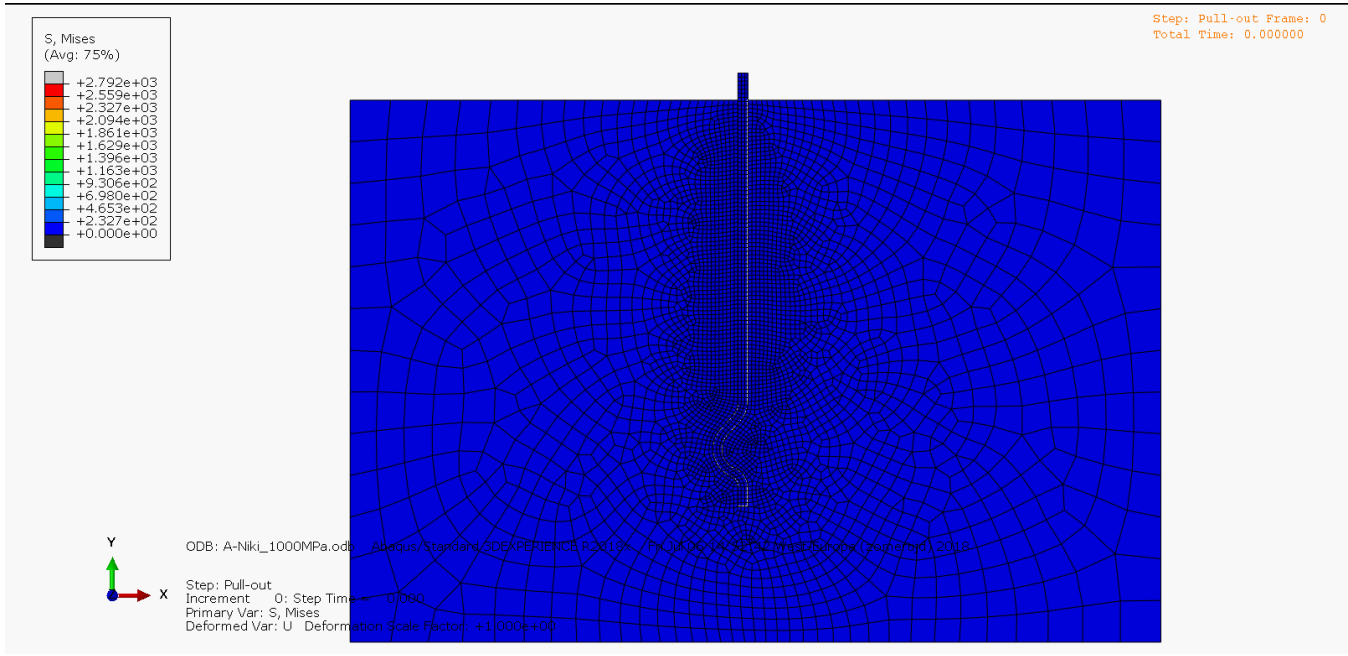
1. Design of collision protection post



ABAQUS at ABT

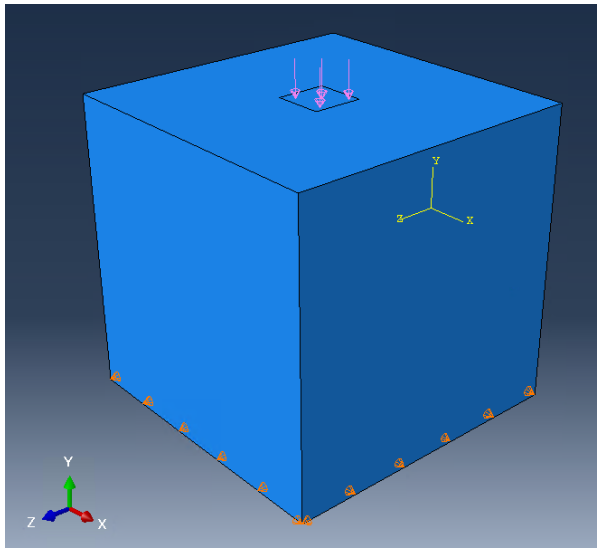
2. Optimization of steel fibre shape for SFRC

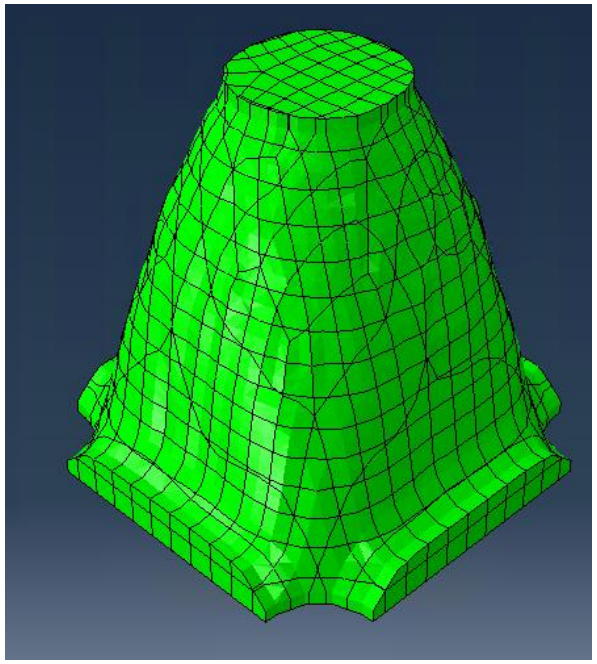


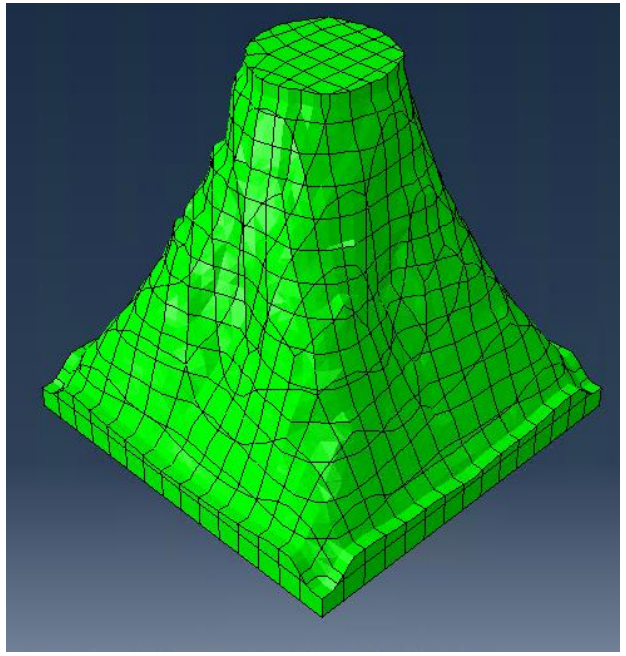


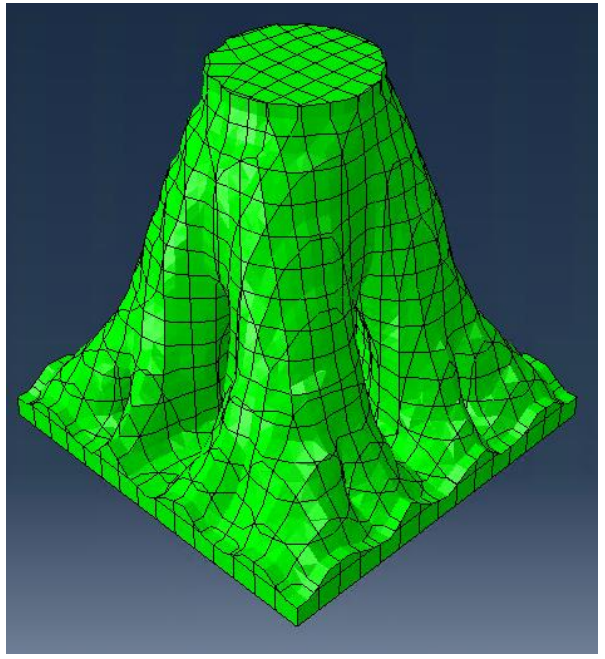
ABAQUS at ABT

3. ‘Natural’ shaped pile cap: shape optimization









Overview of experiments at TU Delft

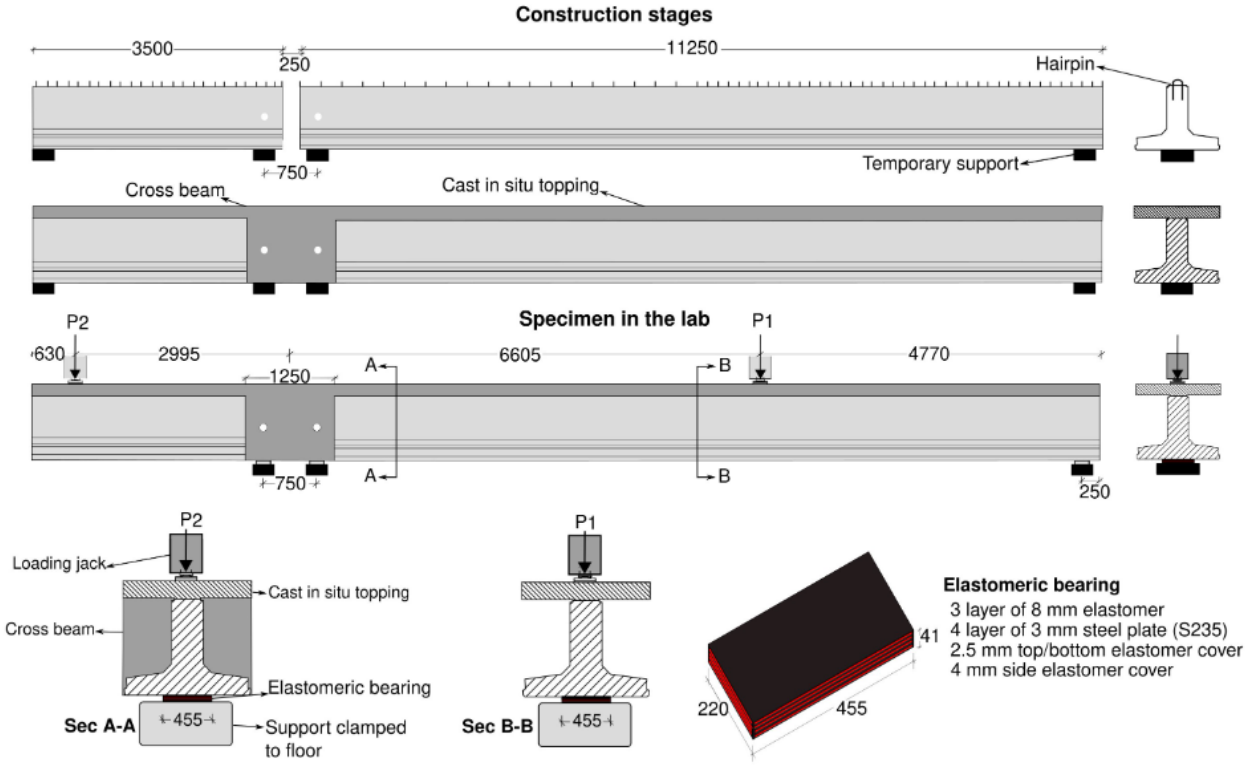
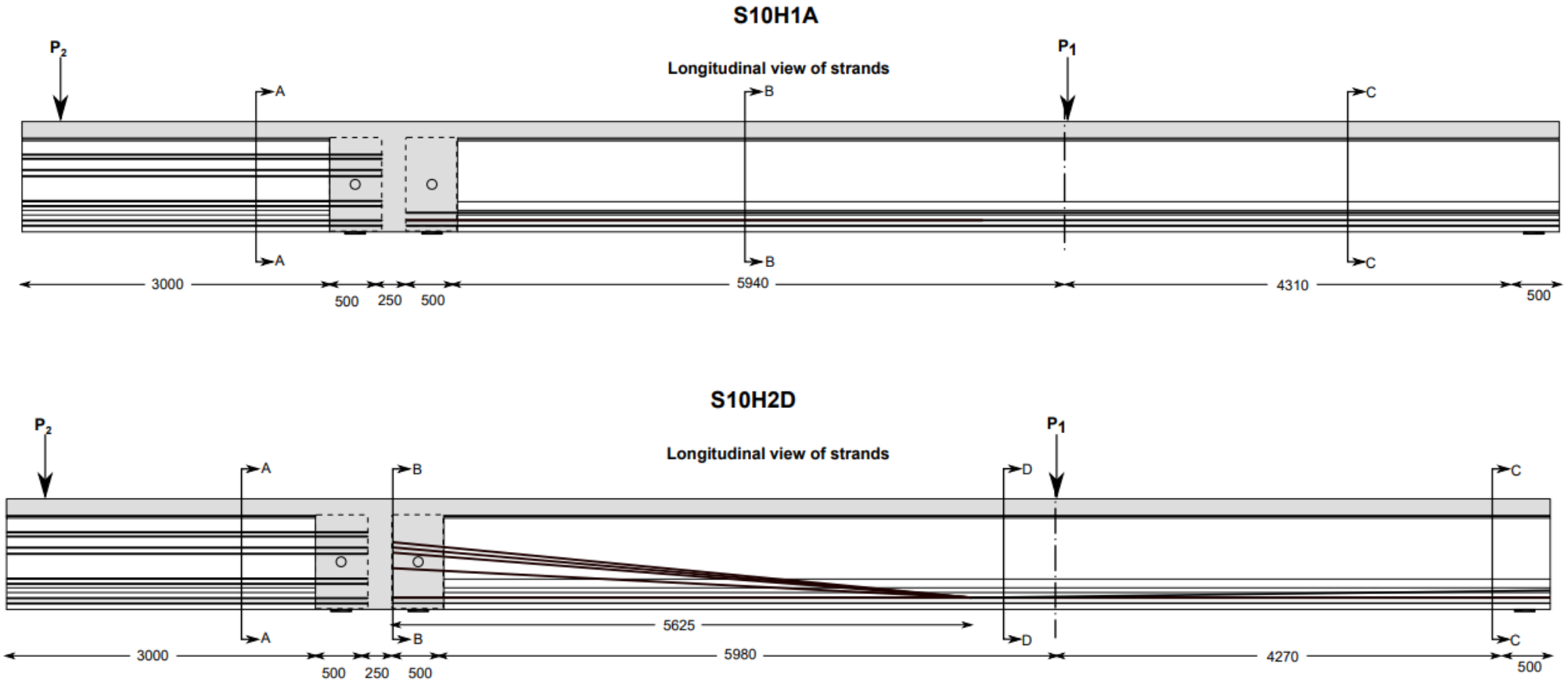


Figure 1: Construction stages of the specimens (All dimensions are in mm)

Figure 2: Interface treatment and hairpin reinforcement

Overview of experiments at TU Delft



Overview of experiments at TU Delft

Figure 3- Specimens in the lab

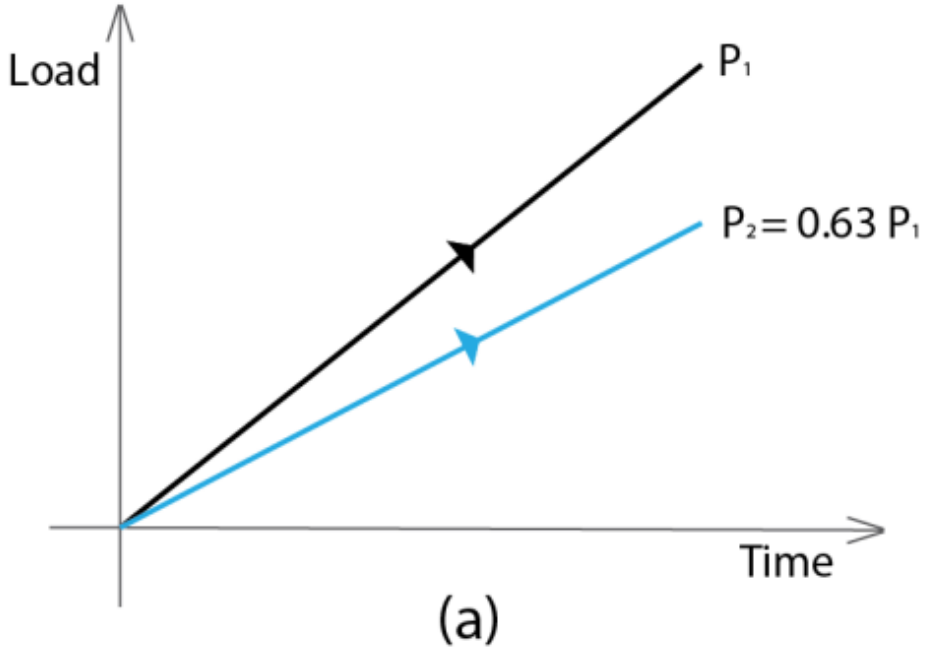
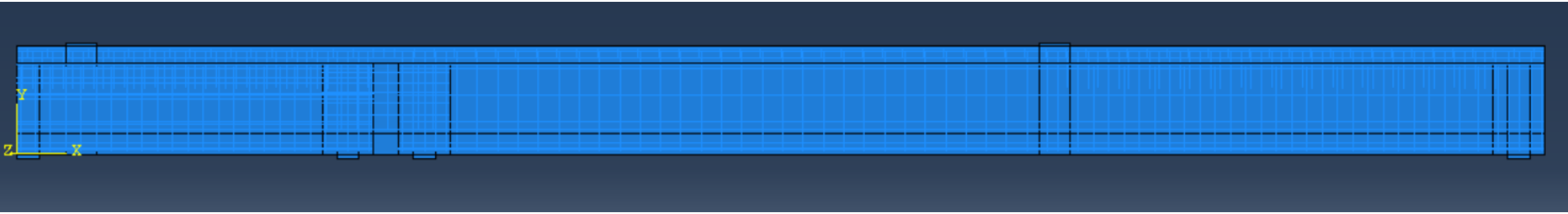
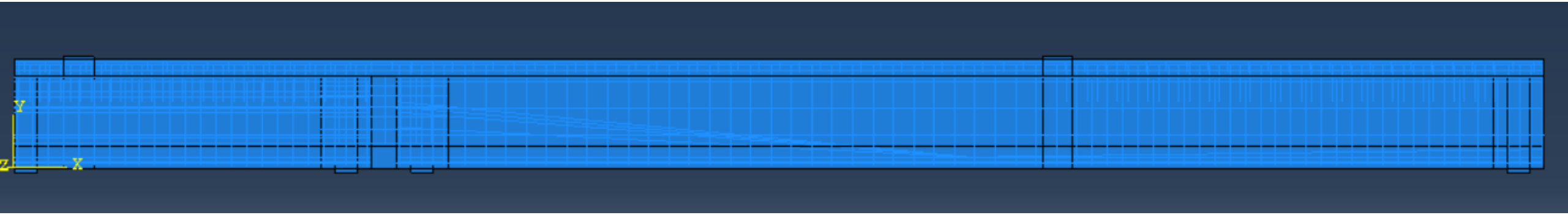


Figure 5-Loading plan for case (a) without yielding of intermediate support (b) with yielding of intermediate support

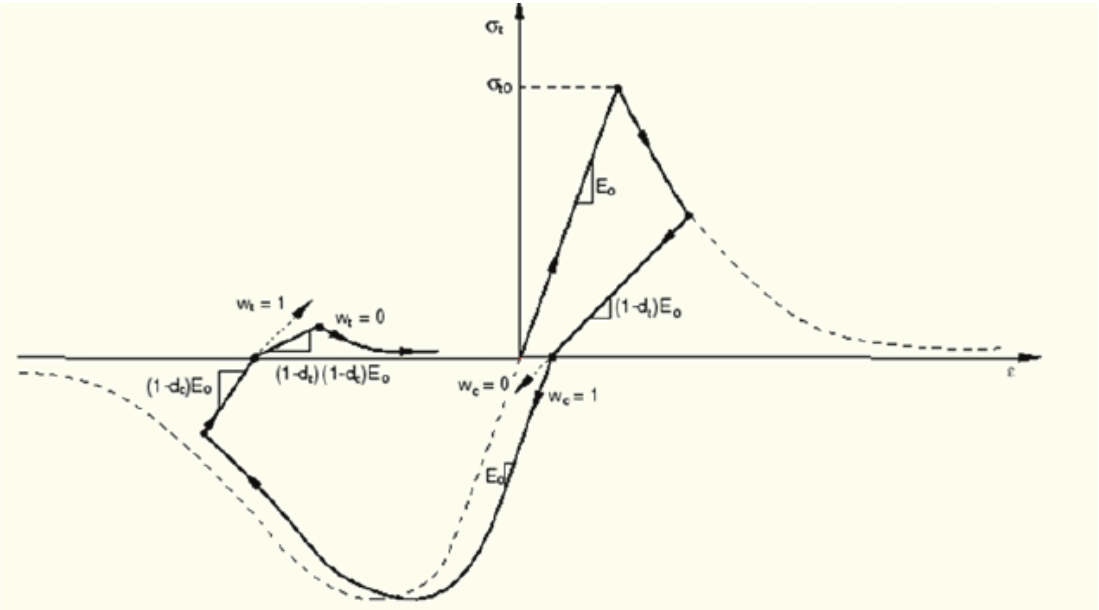
Overview of the ABAQUS model





Overview of the ABAQUS model

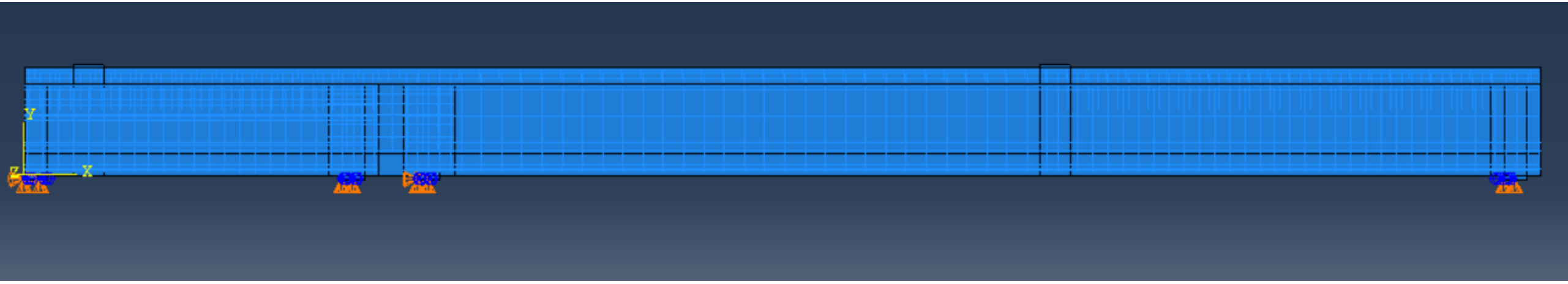
- 2D plane stress model
- Concrete modeled using damage plasticity model



- Reinforcement and prestressing strand was modeled with isotropic plasticity

Overview of the ABAQUS model

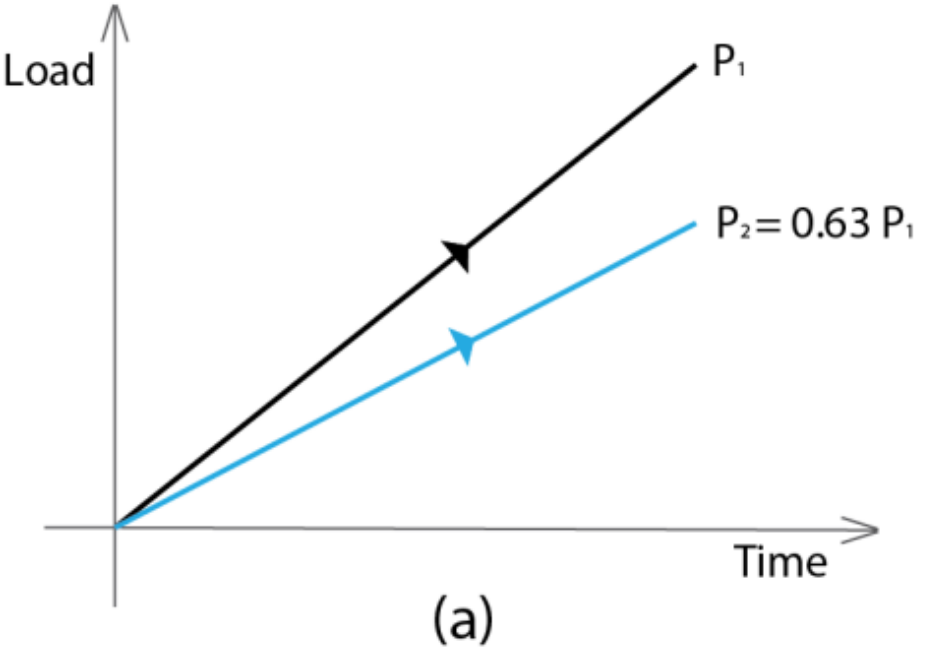
- Contacts:
  - Between the support/loading plates and the beam → ties
  - Between the beam and the topping → Surface-to-surface contact using cohesive contact
- Embedded reinforcement
- Boundary conditions:

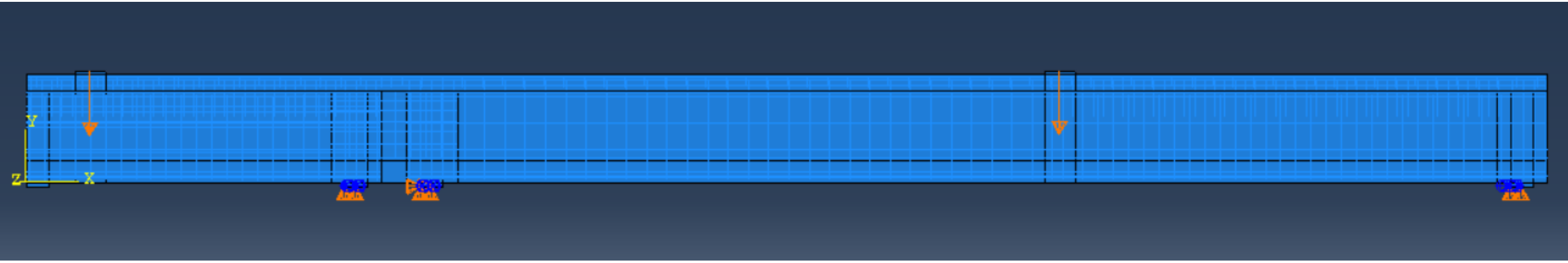


Overview of the ABAQUS model

Loading applied as a phased analysis:

1. Prestress is applied as a predefined field
2. Self-weight of beams
3. Cross beam (skin) (model change)
4. Self-weight of the cross beam
5. Topping (model change)
6. Self-weight of the topping
7. Loading plates (model change)
8. Left support is deactivated
9. Displacement controlled loading

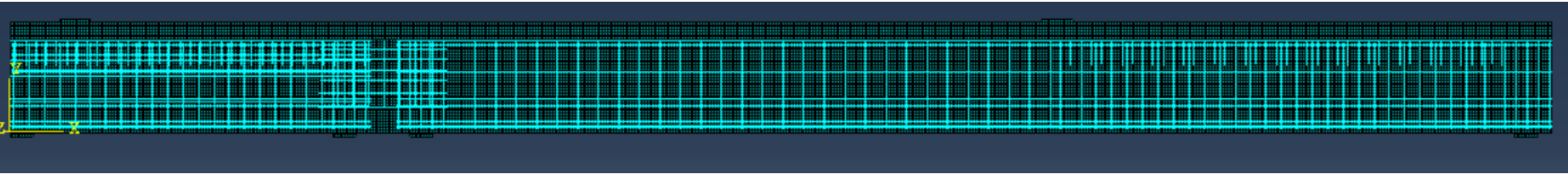




Overview of the ABAQUS model

Mesh:

- Linear quadrilateral elements
- Plane stress elements with reduced integration (CPS4, CPS3)
- Small mesh size for crack localization (~25 mm)



Experimental results: S10H1A

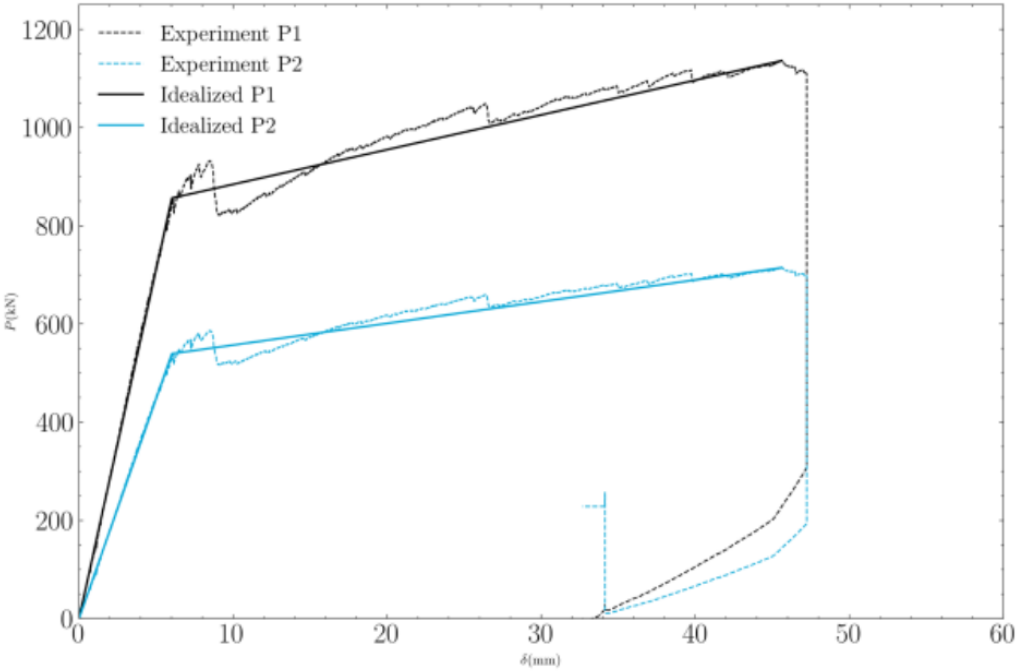
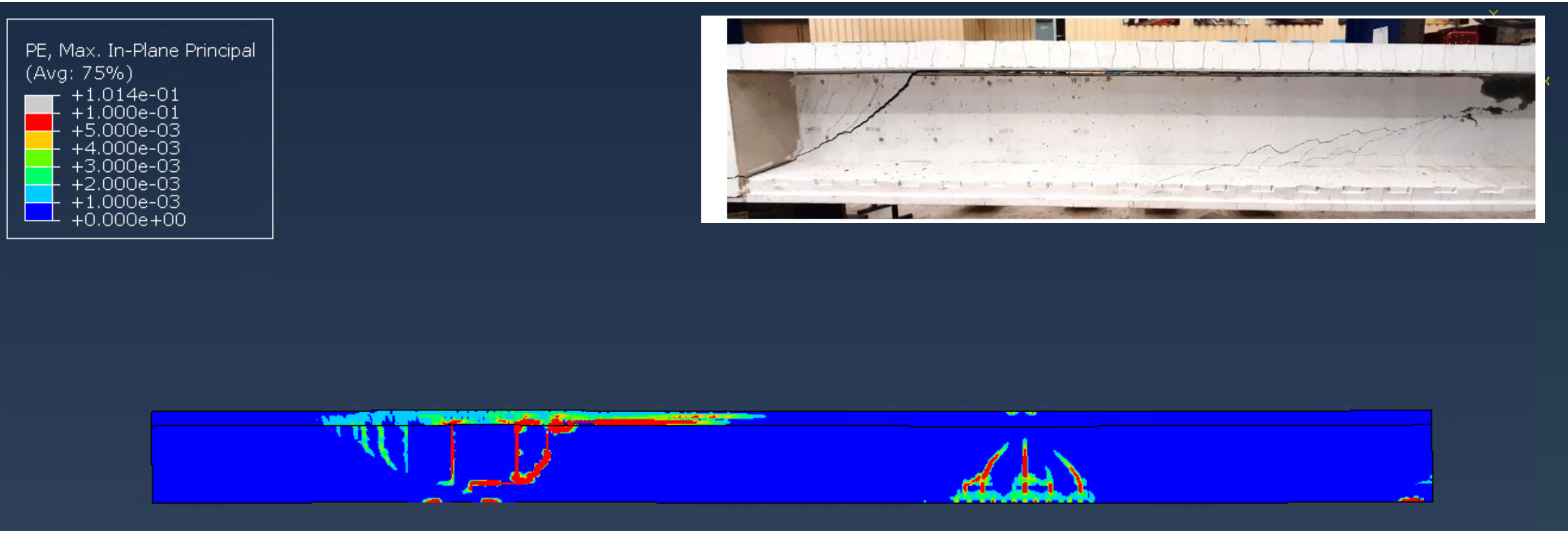
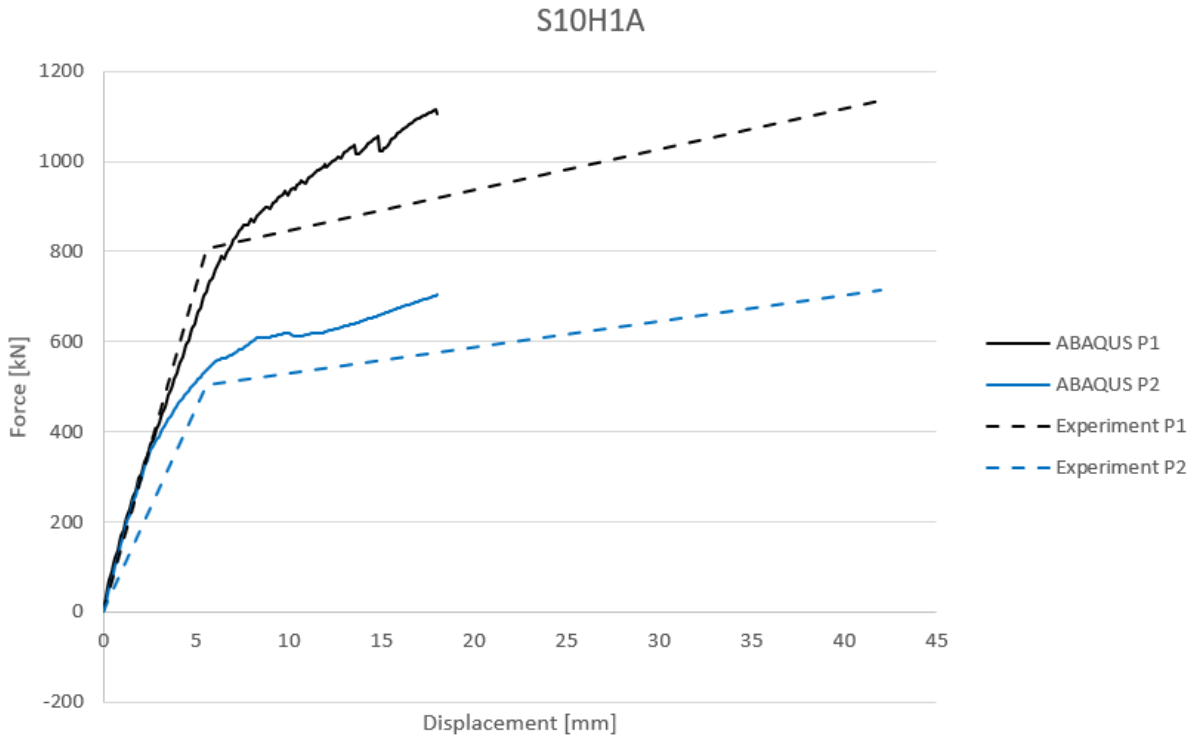


Figure 2: Idealized load-deflection curve for compression failure of specimen S10H1A

S10H1A: crack pattern at maximum load

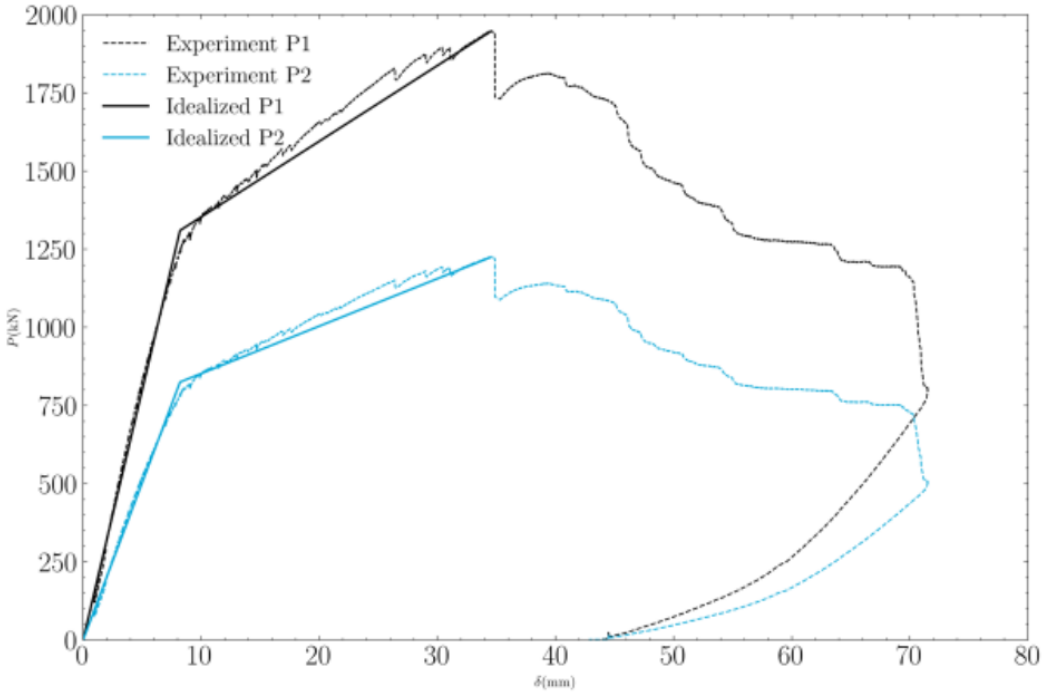


S10H1A: load-displacement curve

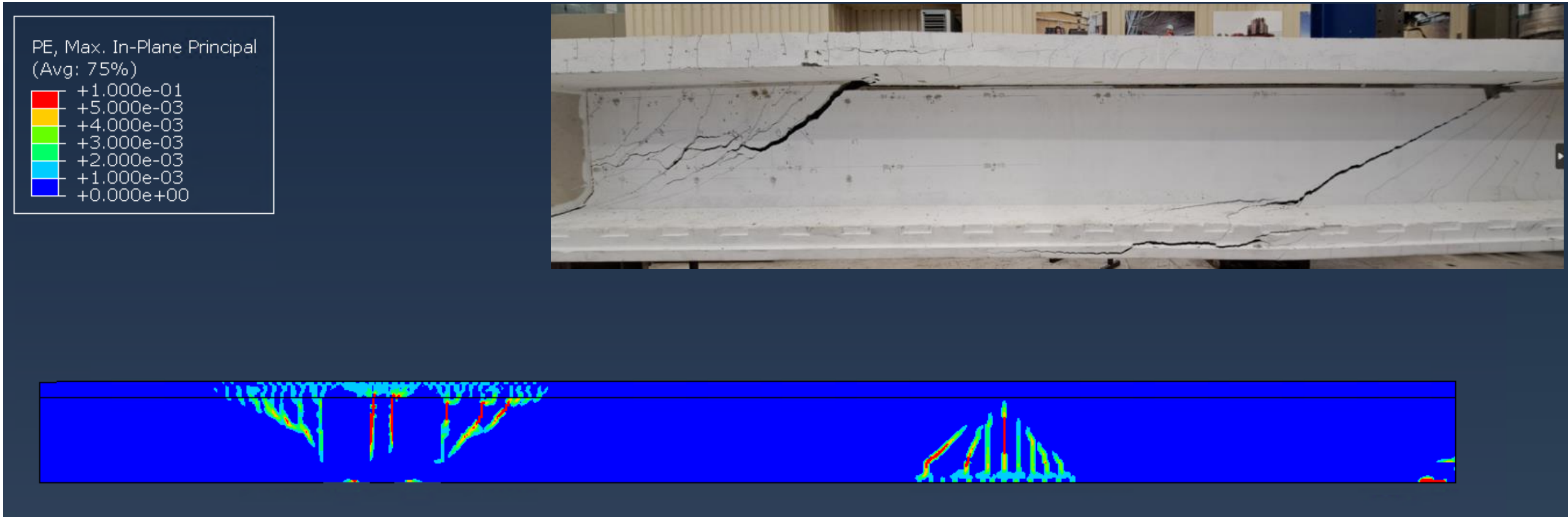


	ABAQUS S10H1A	Experiment S10H1A
P 1,failure [kN]	1105	1136
P 1,cr [kN]	762	856
P 2,failure [kN]	705	714
δ 1,failure [mm]	17,9	45,6
δ 1,cr [mm]	5,9	6,0

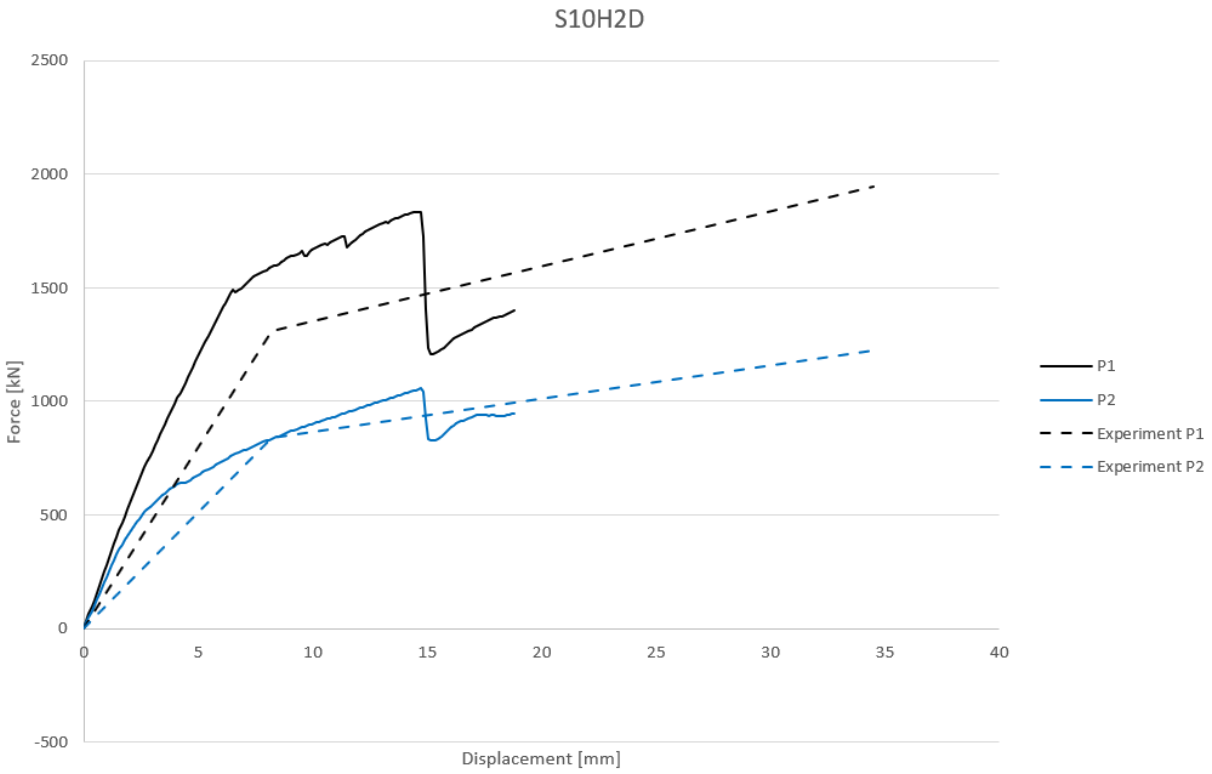
Experimental results: S10H2D



S10H2D: crack pattern at maximum load



S10H2D: load-displacement curve



	ABAQUS S10H2D	Experiment S10H2D
P 1,failure [kN]	1834	1947
P 1,cr [kN]	1478	1310
P 2,failure [kN]	1050	1225
δ 1,failure [mm]	18,7	34,5
δ 1,cr [mm]	11,0	8,2

Future investigation

- 3D model in ABAQUS
- Optimise the model to get a better match in stiffness
- Crack pattern inclination is different

ANY QUESTIONS