SIMULATION OF ROOF PANEL MADE FROM FIBER-REINFORCED CONCRETE

NUMERICAL MODELING AND EXPERIMENTAL TESTING

APPLICANT: VIKTOR FÁBIÁN

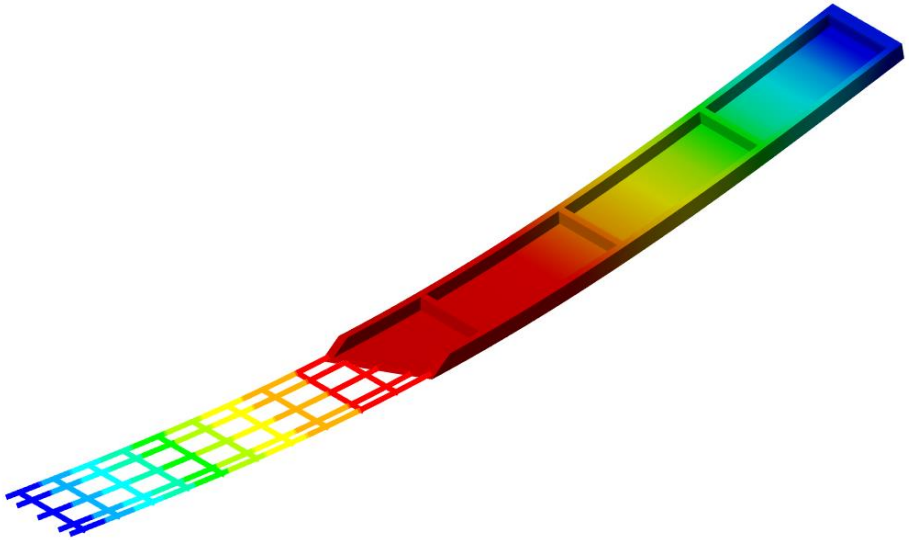
CONSULTANT: DR. ALEKSANDAR LANDOVIĆ

INSTITUTION: UNIVERSITY OF NOVI SAD

FACULTY: FACULTY OF CIVIL ENGINEERING, SUBOTICA

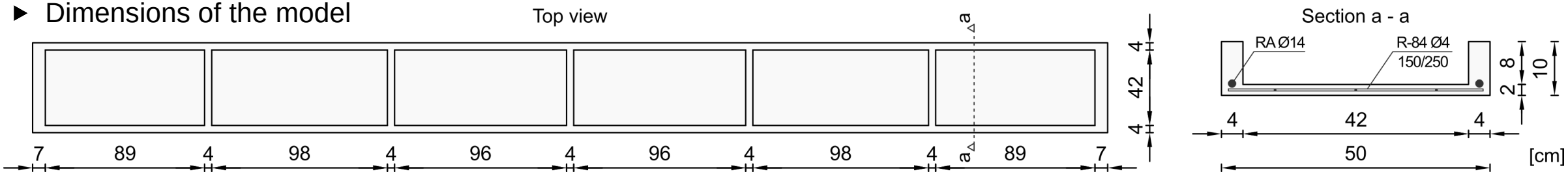
E-MAIL: fabianmviktor@gmail.com

PHONE: +381 63 159 3594



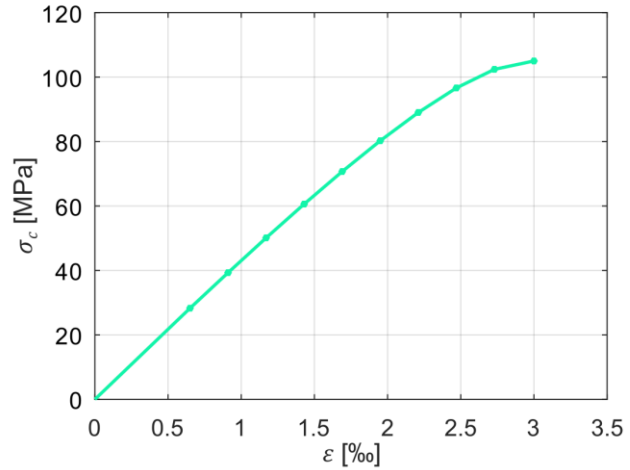
ABOUT THE PROJECT

- Investigation of the load-bearing capacity of a roof panel made from fiber-reinforced concrete using ANSYS Mechanical software
- Dimensions of the model



- Mechanical properties of the materials used to simulate the behaviour of the roof panels

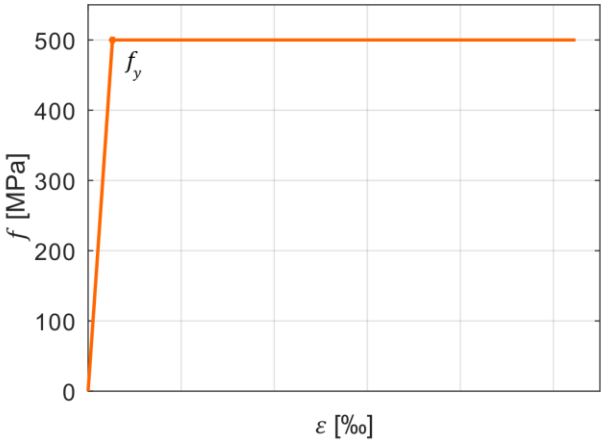
ε	σ_c
0.00065	28.289
0.00091	39.316
0.00117	50.111
0.00143	60.606
0.00169	70.701
0.00195	80.250
0.00221	89.020
0.00247	96.632
0.00273	102.404
0.00300	105.000



$$\sigma_c = -f_{cm}\left[\frac{k\eta - \eta^2}{1 + (k-2)\eta}\right]$$

REINFORCEMENT	
f_y	500 [MPa]
E	200 [GPa]
μ	0.2

CONCRETE	
f_{cm}	105 [MPa]
f_{tm}	7.96 [MPa]
E	43.52 [GPa]
μ	0.2



- Acquiring mechanical properties through experiments - concrete samples in the laboratory

NUMERICAL MODEL SCHEMATIC AND DISCRETISATION

- Static structural analysis

LINEAR ANALYSIS (LI)

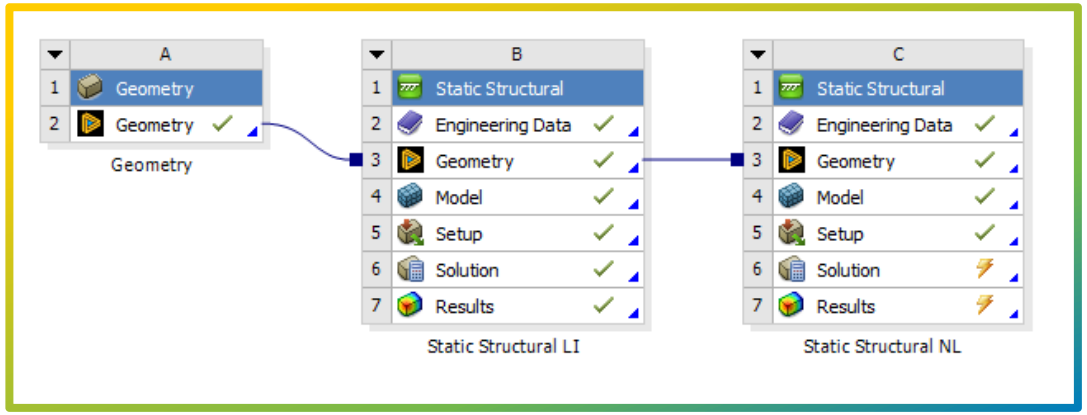
for calculating small deformations where material behavior remains linear

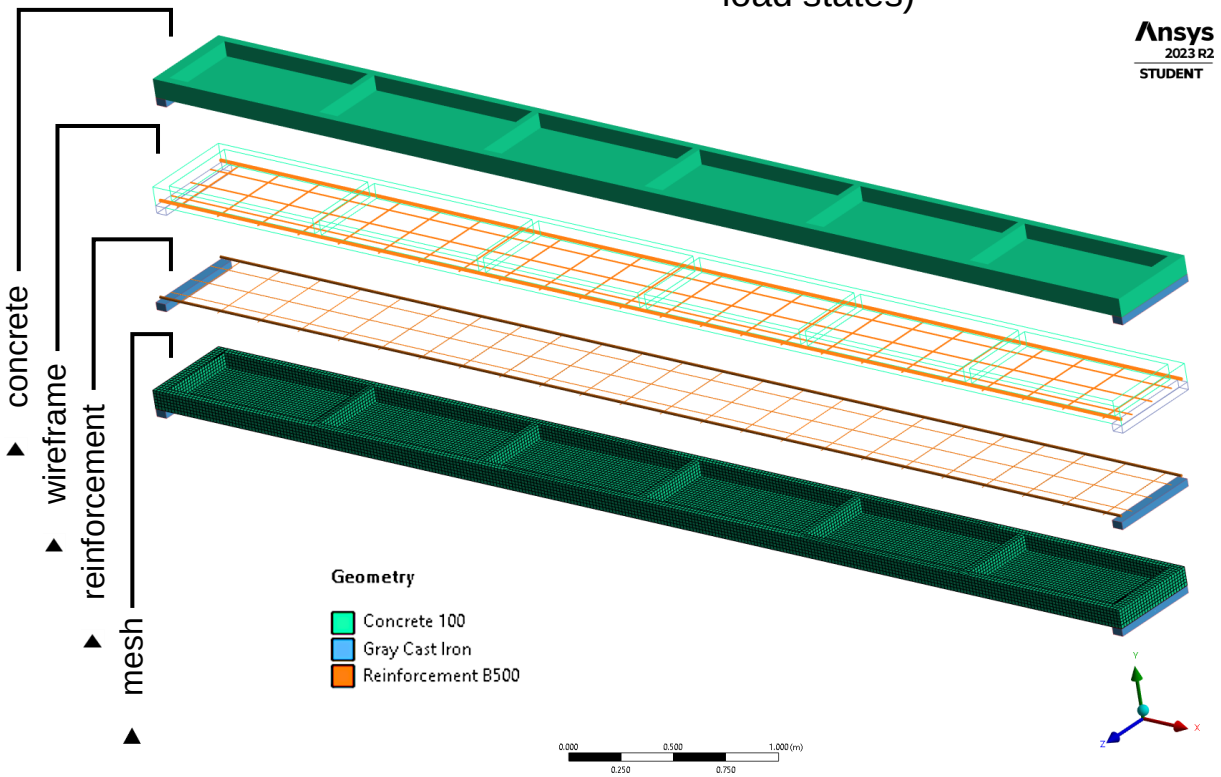
(used for exploitation load level)

NON-LINEAR ANALYSIS (NL)

gives a more accurate behaviour considering large deformations

(used for calculation for ultimate load states)



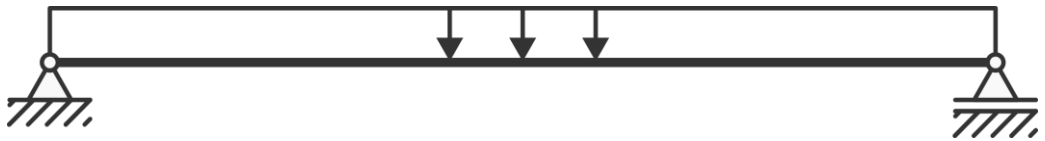


- Mesh statistics: the element order is *linear*, and the element size is 0.02 [m]

  Concrete / *stiffness behaviour – flexible* / 16014 elements

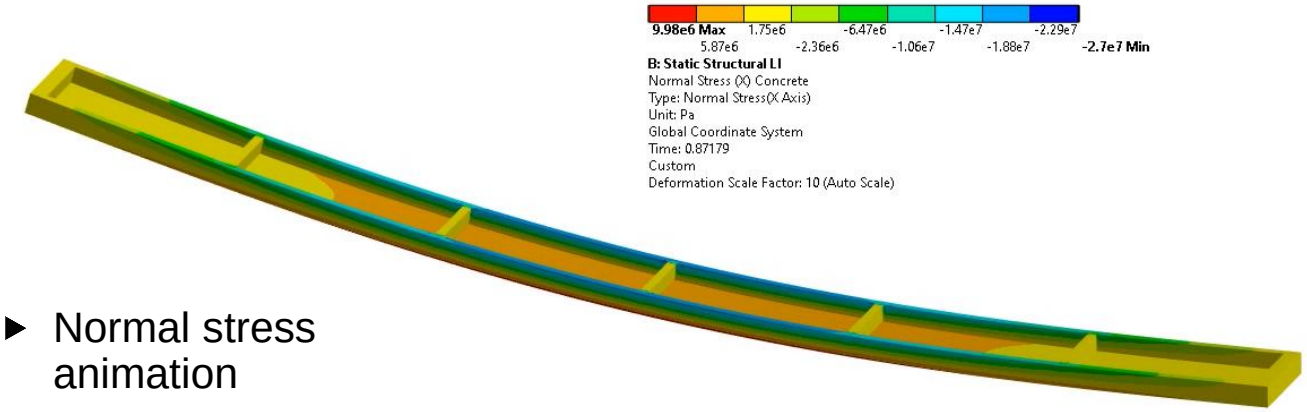
  Rebars / *model type – reinforcement* / 2071 elements

  Supports / *stiffness behaviour – rigid* / 152 elements

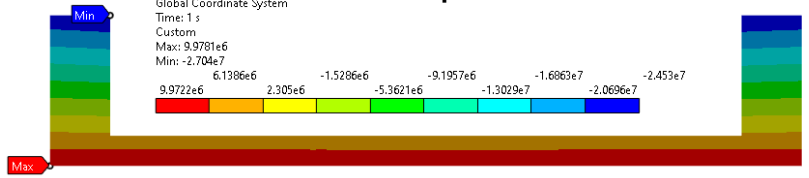


- The supports are defined as *Remote Displacements* – one pinned and one roller support, with *bonded* contact zones between the supports and the concrete panel

LINEAR ANALYSIS - RESULTS

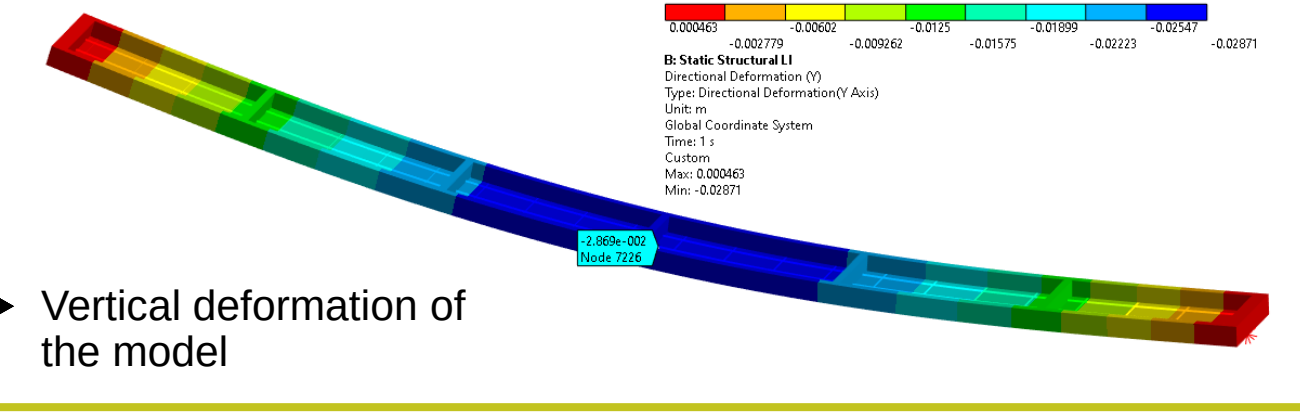


- Normal stress animation

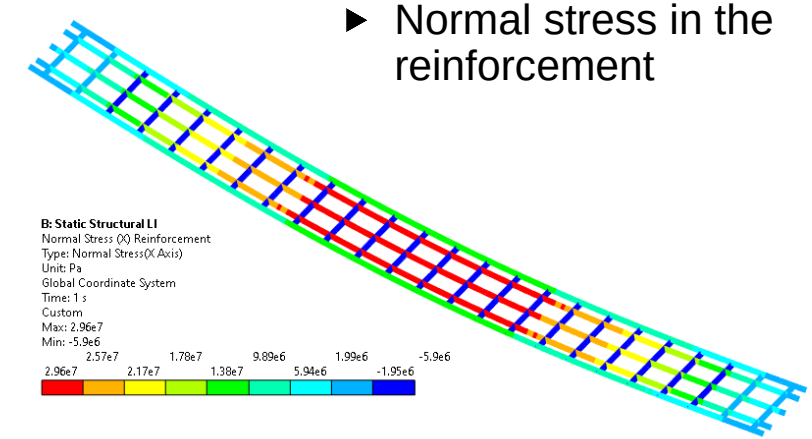


- Normal stress in the midpoint cross-section

- The applied load consists of: self-weight of the panel 950 kN/m^2 + additional load (load from the first phase of the experiment) 1335 kN/m^2 = 2285 kN/m^2
- Vertical displacement of the model, obtained with the given uniform load is around 28.7 mm

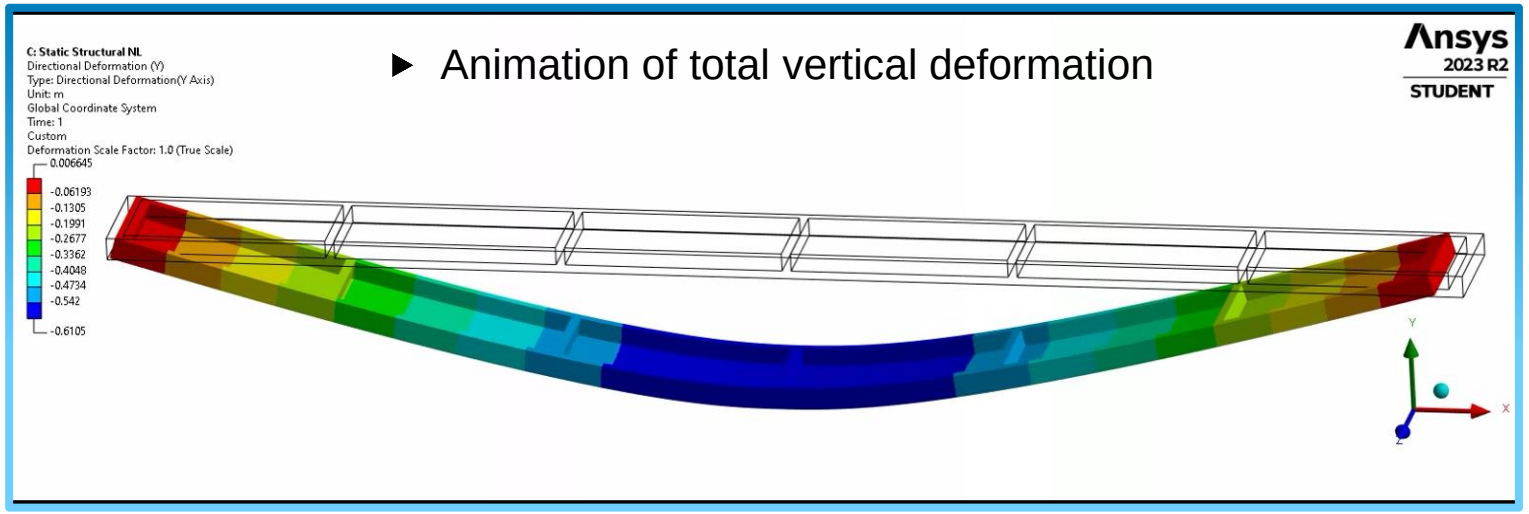


- Vertical deformation of the model

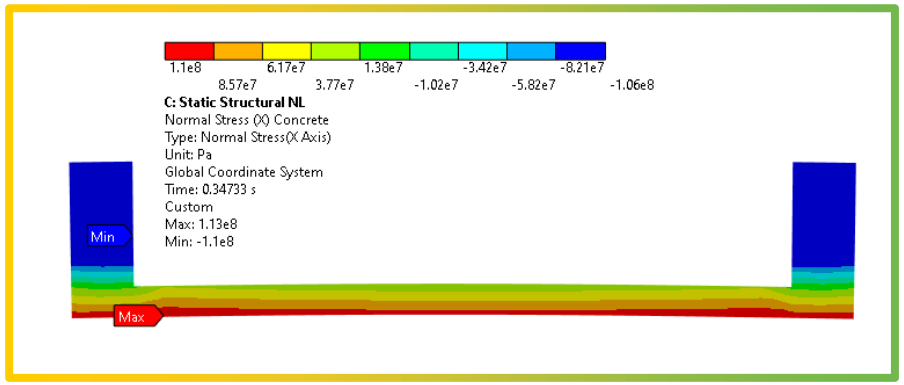


- Normal stress in the reinforcement

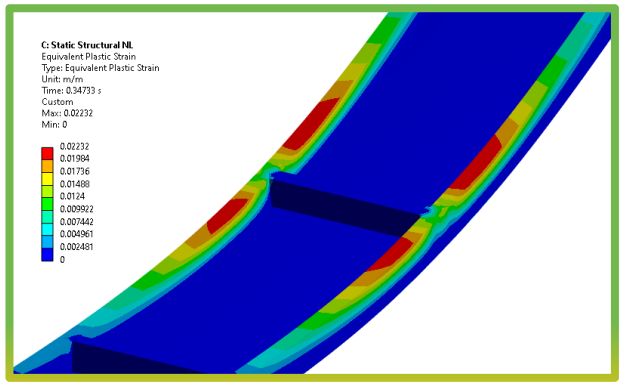
NONLINEAR ANALYSIS - RESULTS



- Animation of total vertical deformation



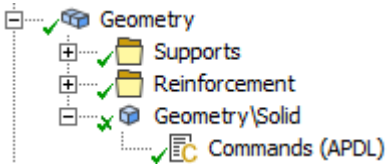
- Normal stress in the midpoint cross-section

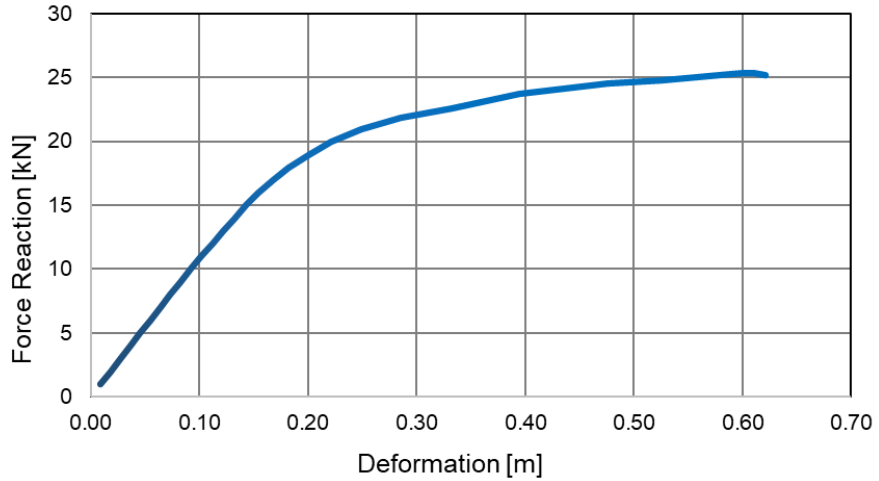


- Plastic strain

Step Controls	
Number Of Steps	1.
Current Step Number	1.
Step End Time	1. s
Auto Time Stepping	On
Define By	Substeps
Initial Substeps	75.
Minimum Substeps	75.
Maximum Substeps	1000.
Solver Controls	
Solver Type	Program Controlled
Weak Springs	Off
Solver Pivot Checking	Program Controlled
Large Deflection	On
Inertia Relief	Off
Quasi-Static Solution	Off

- Details of the analysis settings
- Mechanical properties of the concrete were assigned using the *APDL* command





- Relation between midspan displacement and reaction force

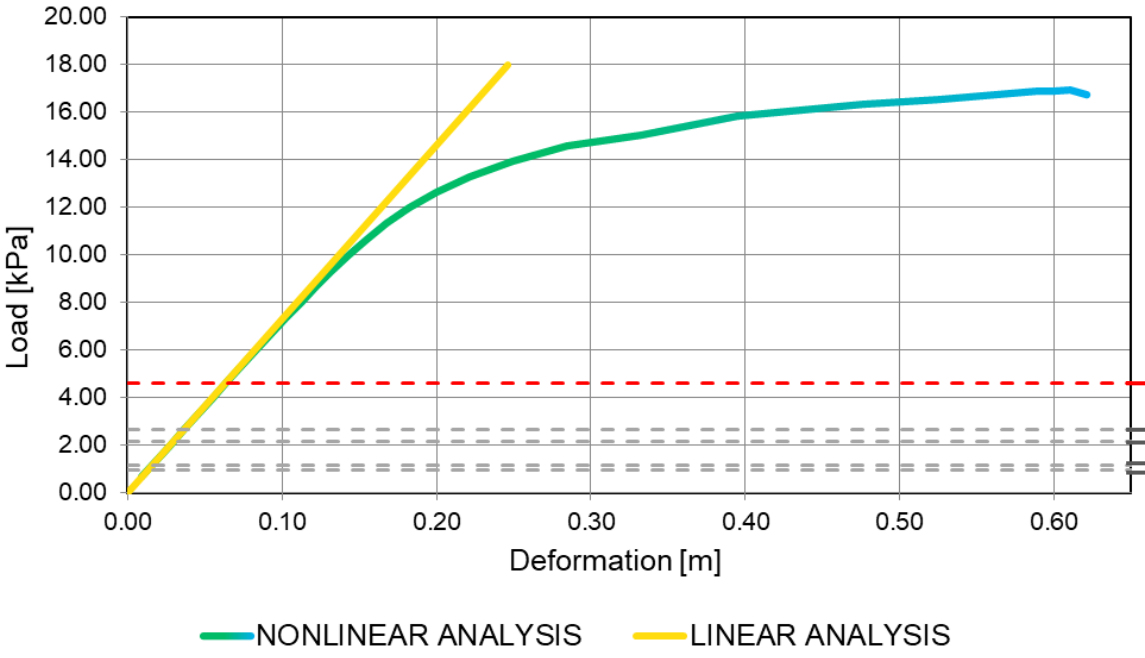
NUMERICAL ANALYSIS VS. EXPERIMENT

ADVANTAGES OF NUMERICAL ANALYSIS:

- with lineal analysis, we can confidently calculate the forces that appear in the structure for different loads
- a nonlinear analysis is necessary to obtain the load-bearing capacity and potential structural behavior at failure

EXPERIMENT:

- experiments provide the most accurate results but only if they're done correctly
- but unfortunately they are often more time consuming, financially exhausting and involve risks



maximal load during experiments

wind load + s.w. + r.c. + s.

snow load (s) + s.w. + r.c.

roof covering (r.c.) + s.w.

self-weight (s.w.)

} Loads on the roof panel during exploitation

CONCLUSIONS

- To obtain accurate results from numerical analysis, it is necessary to:
  - use the appropriate material models – ensuring the correct behavior of materials such as concrete and steel
  - select the correct finite element types – choosing the right modelling elements that are compatible with each other
  - choose good boundary conditions and support connections – describing the real-life conditions accurately

- Thanks to the ANSYS Mechanical software, we can achieve results that may not be attainable with experimental methods due to the physical limitations and/or safety issues
- It can be concluded that the software is capable of accurately describing the complex behavior of the fiber-reinforced concrete roof panel, providing us with valuable insights into its performance which would be more difficult to achieve through experimentation alone
- Furthermore, an accurate numerical analysis also eliminates the human factor, thus avoiding potential errors that may occur during experimental testing