Support for applications in fire design

Benchmark studies on composite floors and composite columns subjected to elevated temperature

ing. Ioan Both, PhD.

Department of Steel Structures and Structural Mechanics

Overview

Introduction

Reference case

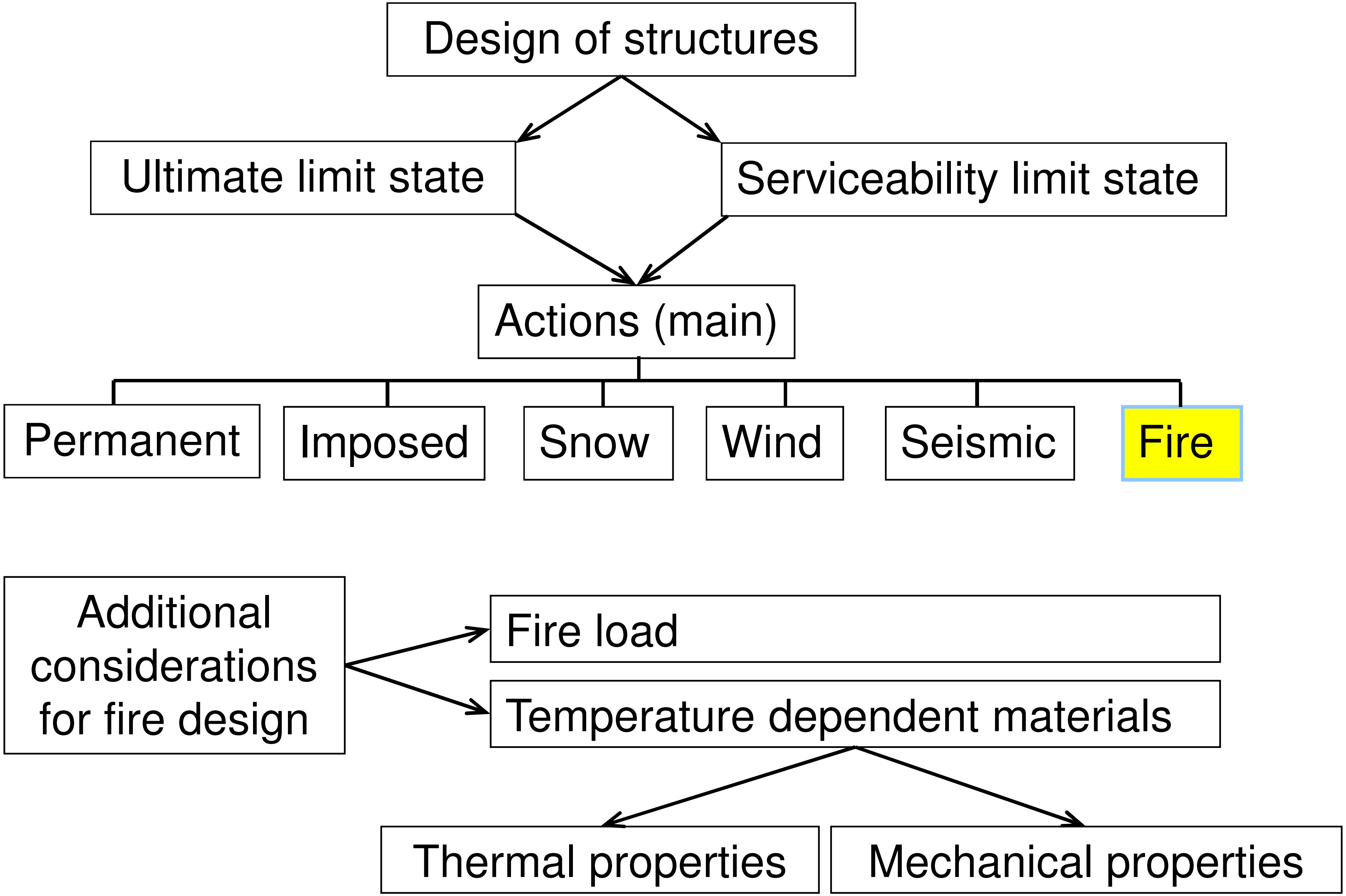
Validation

Benchmark model

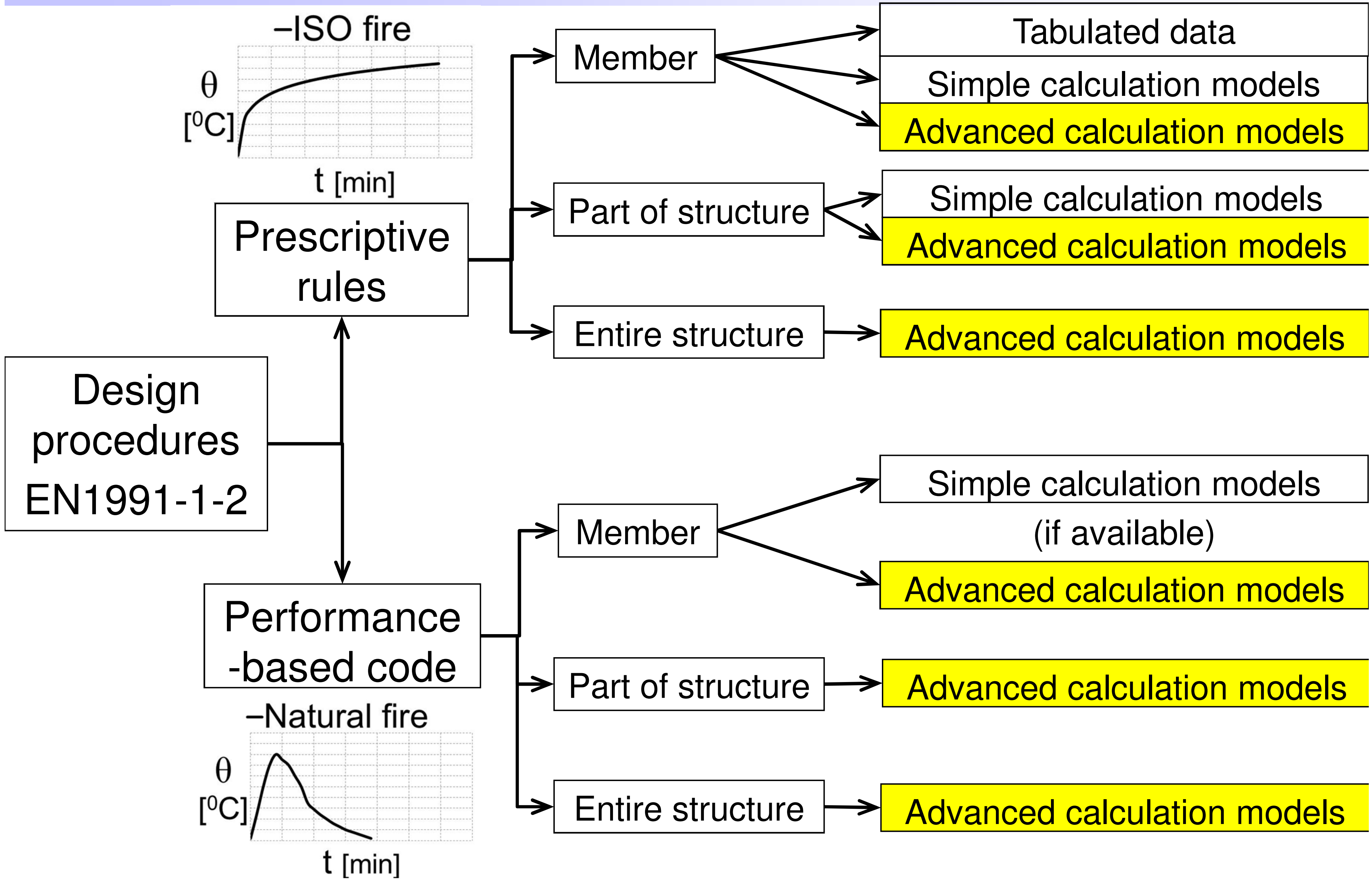
- Input data
- Numerical model details
- Results

Conclusions

Introduction



Introduction



Introduction

Advanced calculation models:

- shall provide a realistic analysis of structures exposed to fire. They shall be based on fundamental physical behavior in such a way as to lead to a reliable approximation of

the expected behavior of the relevant structural component under fire conditions

Section 4.4.4 of EN 1994-1-2

Validation of advanced calculation models

- A verification of the calculation results shall be made on basis of relevant test results
- The critical parameters shall be checked, by means of a sensitivity analysis, to ensure that the model complies with sound engineering principles

Introduction

Advanced calculation models

- Dedicated FE programs

Computational environment is predefined

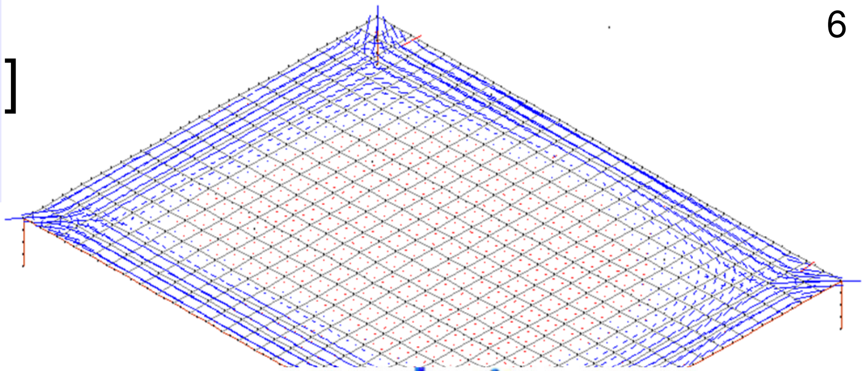
- SAFIR
- VULCAN
- etc.

- General purpose FE programs

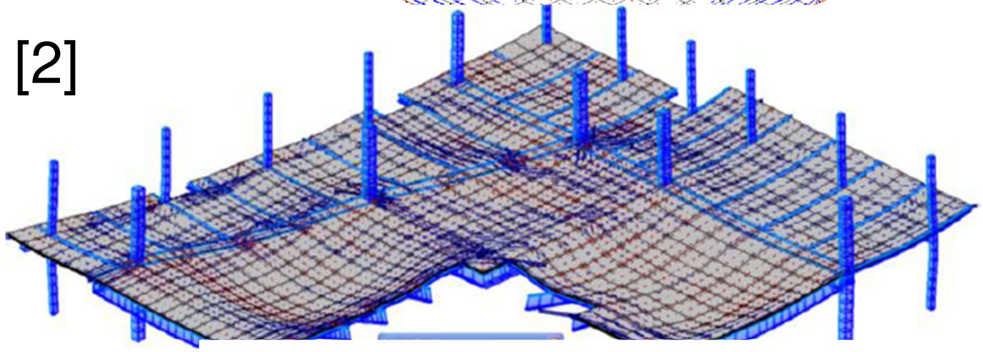
Specific settings should be defined

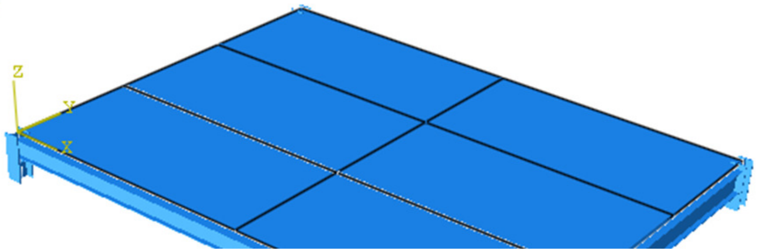
- ABAQUS [4]
- ANSYS
- etc.

[1]

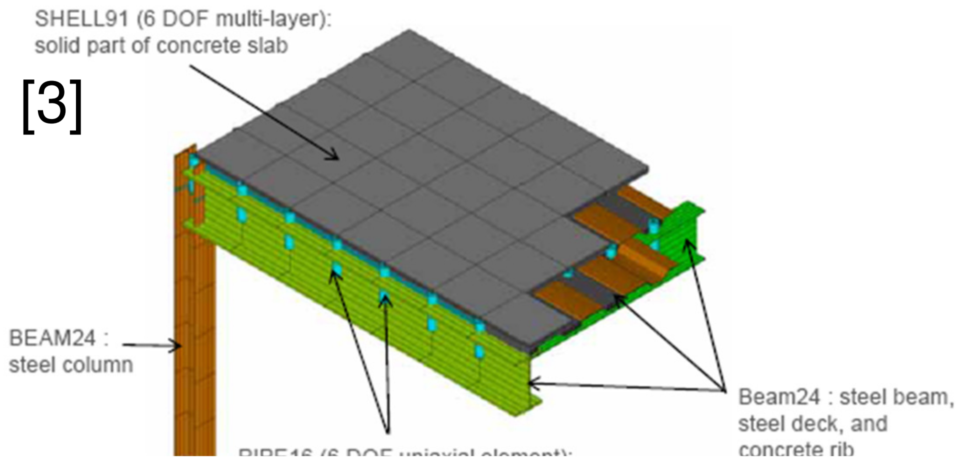


[2]



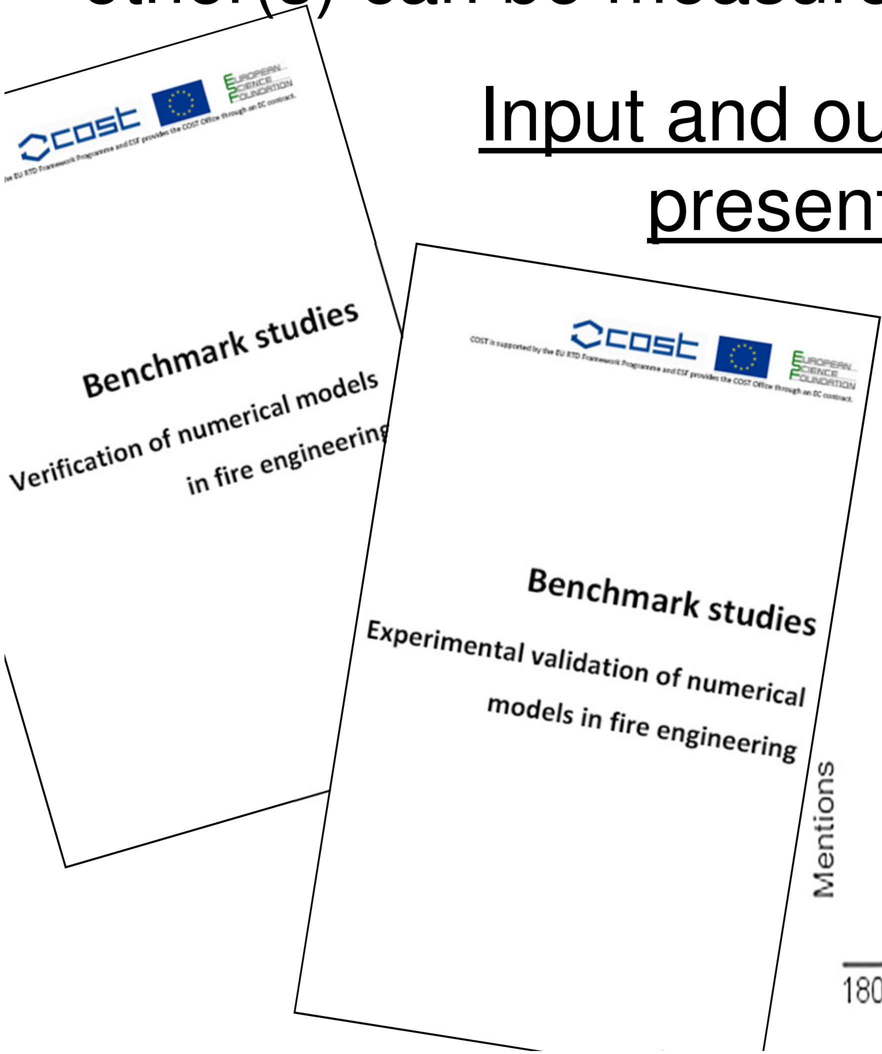


[3]



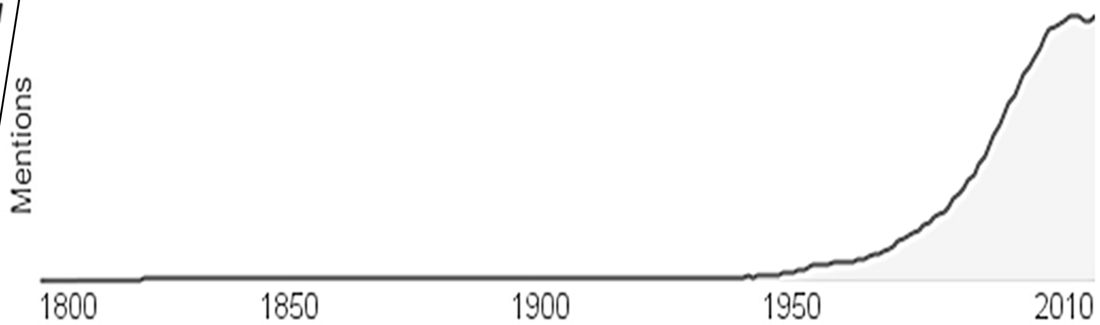
Introduction

Benchmark: a standard or reference by which other(s) can be measured or judged [5]



Input and output data should be presented in detail [6]

Number of "benchmark" use [5]



Introduction

Reference case
- complex input data

Validate a methodology
Compare the results of a complex model

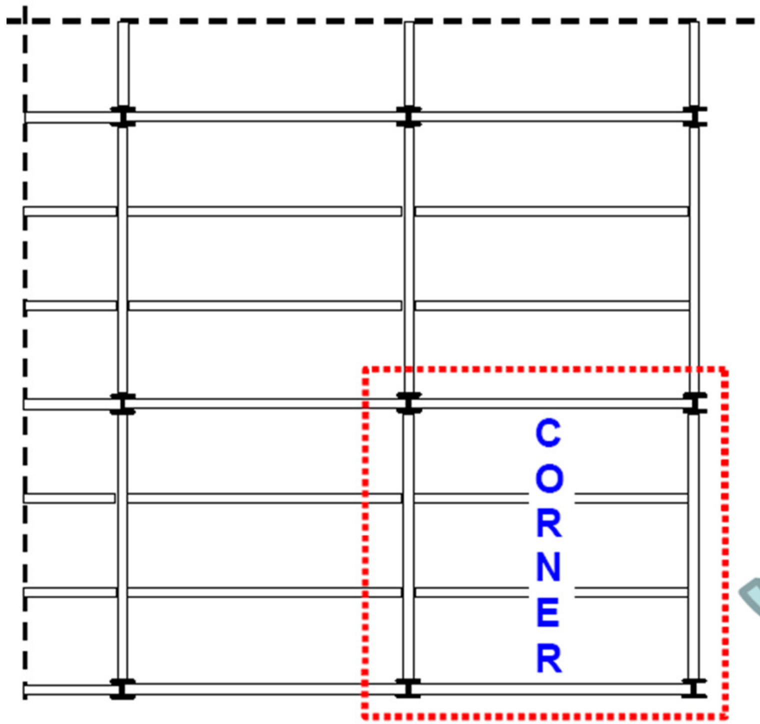
Simplify the input data
- keep the basic phenomena

Describe methodology
- Analysis considerations

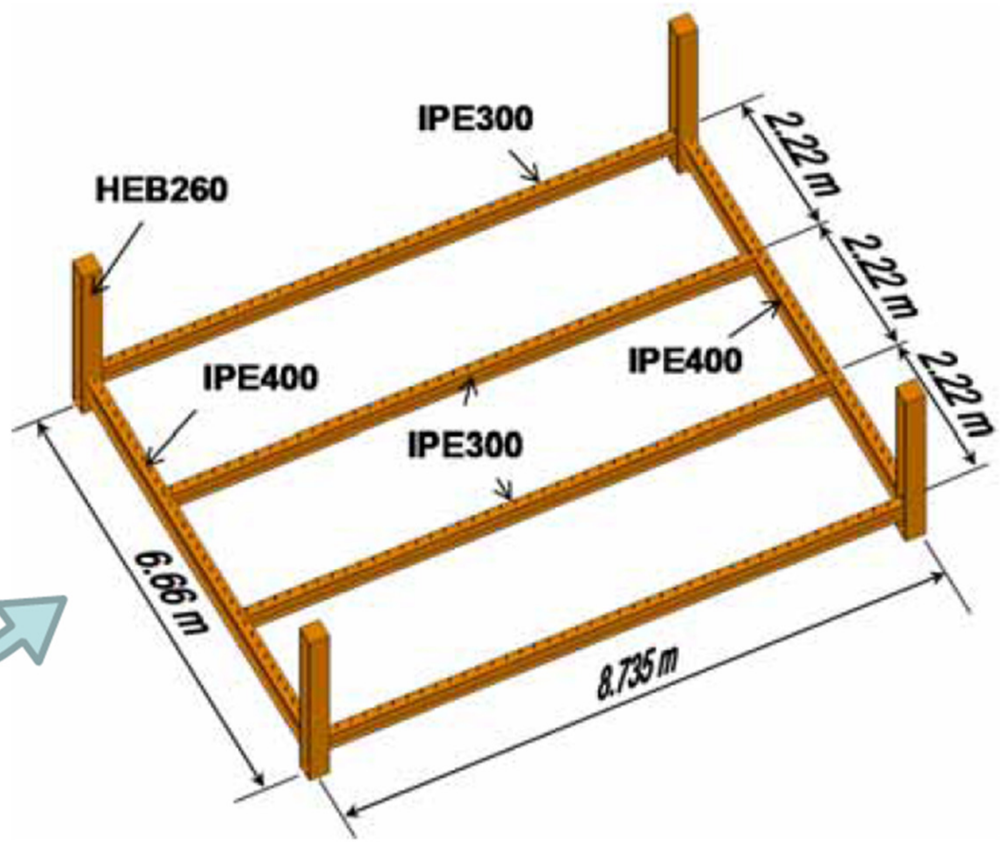
Present the results
- Simplified considerations

Reference case

FRACOF fire test - **Setup**



Grid of a real structure

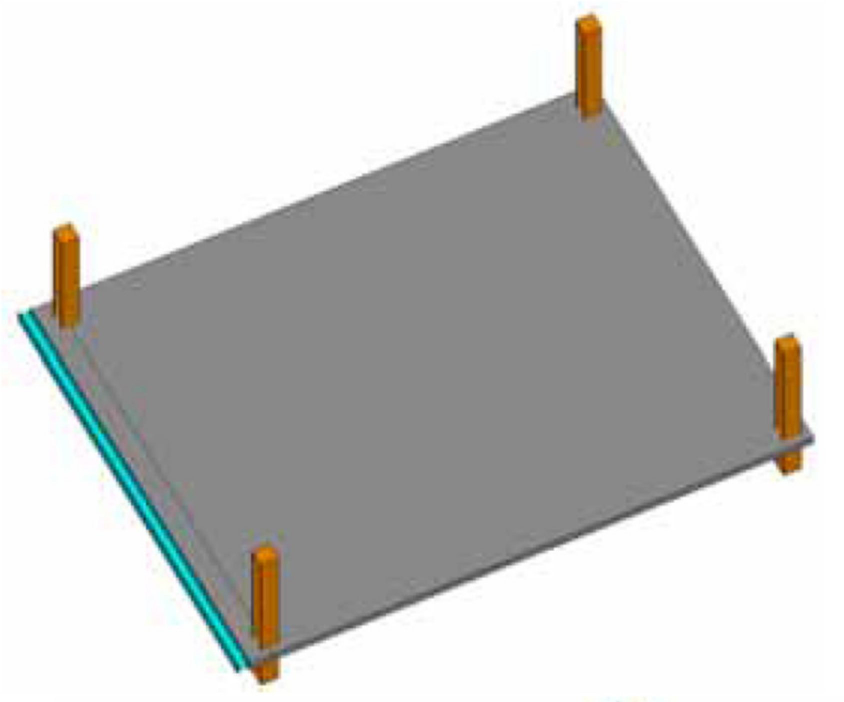


Elements of tested structure

[3]

Reference case

FRACOF fire test - **Setup**



Composite floor

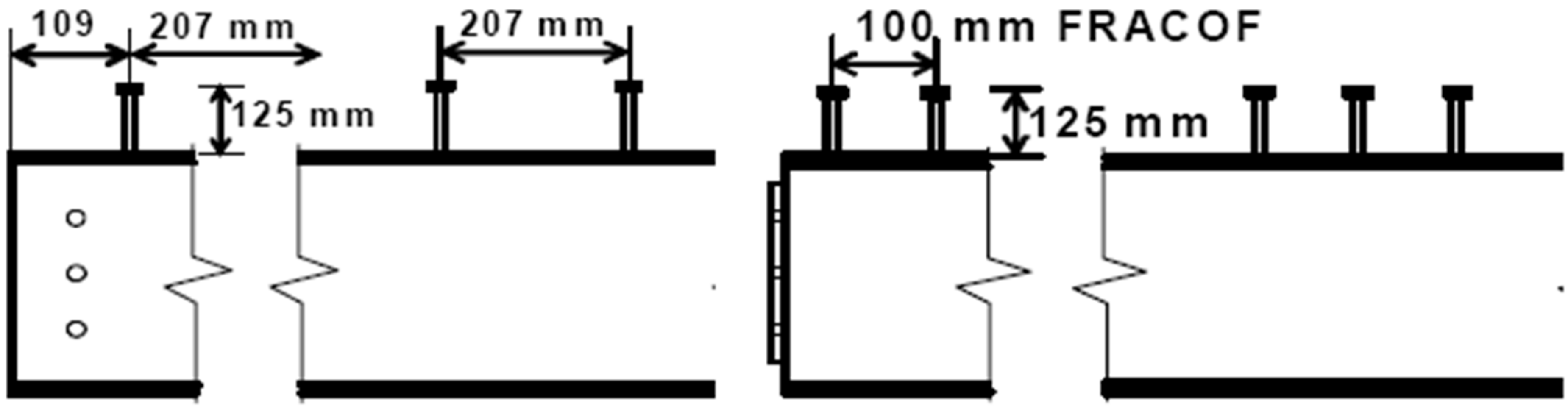
Real-scale specimen

[3]

Reference case

FRACOF fire test

Beam to slab connections



Secondary beams IPE300

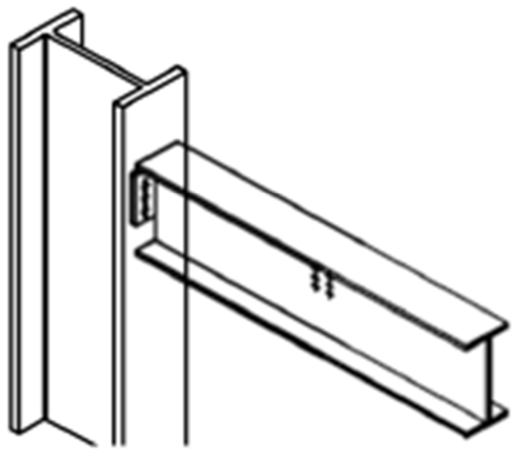
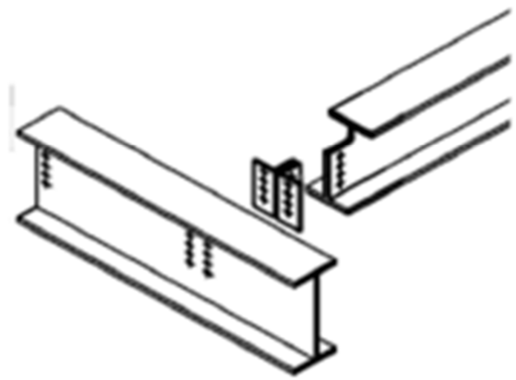
Primary beams IPE400

Full shear connection !

[3]

Reference case

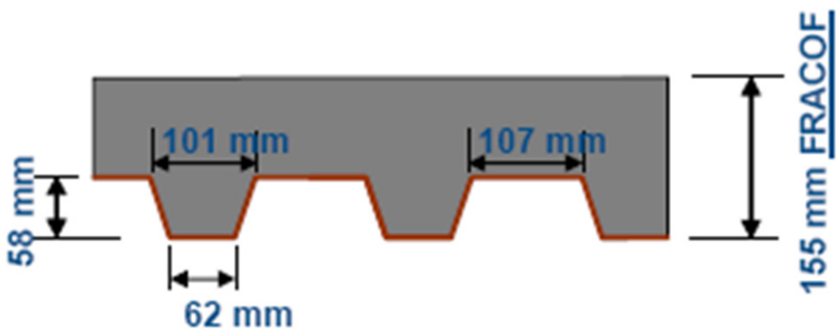
FRACOF fire test - **Connections**

Beam to column		Beam to beam
Secondary beam	Primary beam	
Double angle web cleats	Flexible end plate	Double angle web cleats
		

[3]

Reference case

FRACOF fire test - **Materials**



Structural steel: S235

Reinforcing steel mesh:

S500 Ø7 / 150x150

Axis distance from top of the slab:

50mm

Concrete: C30/37

Steel deck: COFRAPLUS60 - 0.75mm

Secondary beams: f_y=311 N/mm^2

Primary beams: f_y=423 N/mm^2

Reinforcing steel mesh: f_y=594 N/mm^2

Concrete cylinder compressive strength: f_c=36.7 N/mm^2

[7]

Reference case

FRACOF fire test - **Loads**

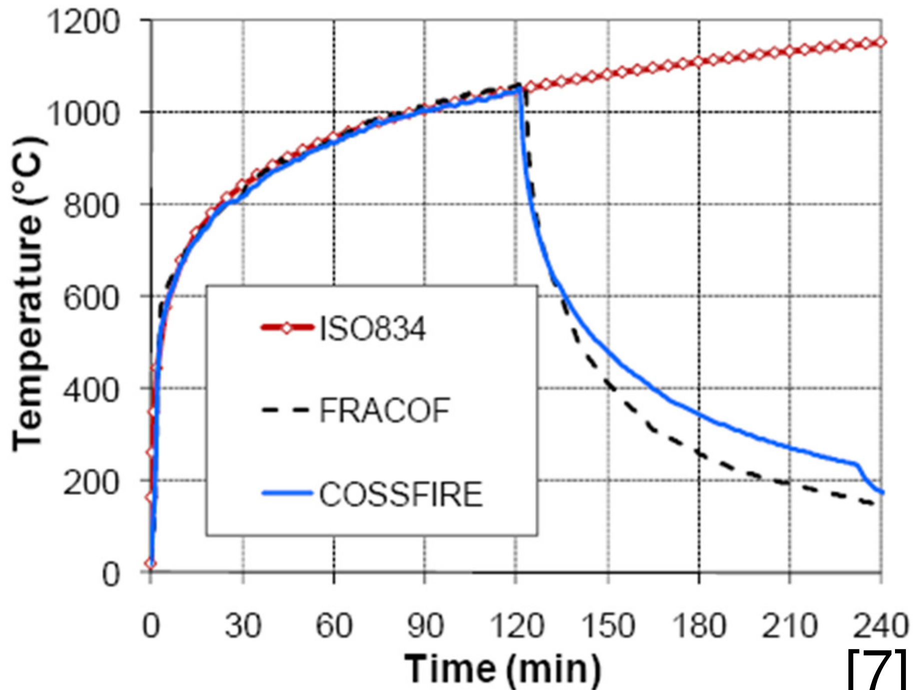
Mechanical load

15 sand bags x 1512 kg

Equivalent uniform load: 390kg/m²

Fire load

120 min of Standard fire curve ISO 834 and a cooling phase

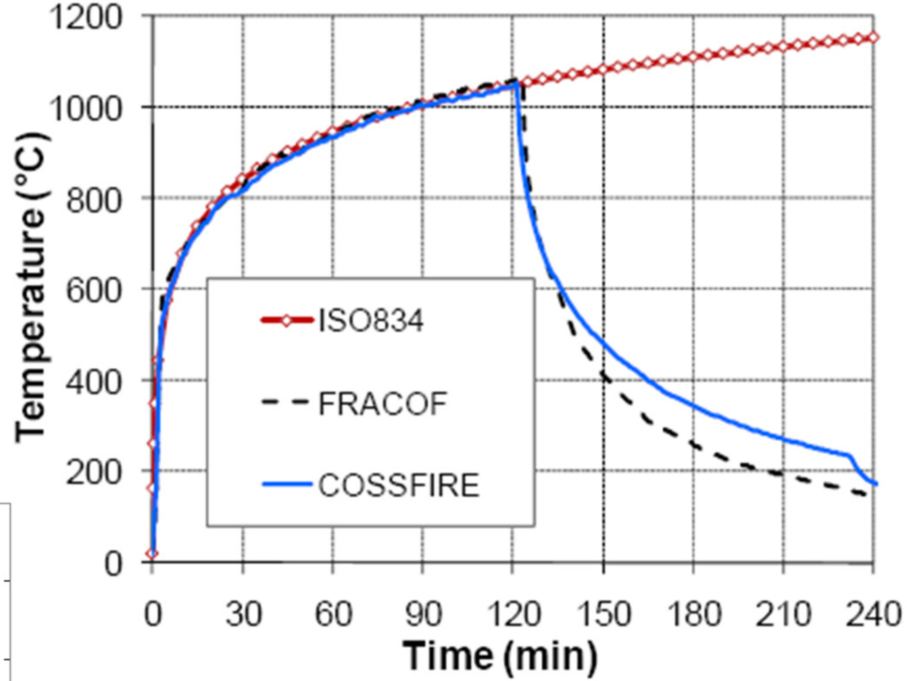


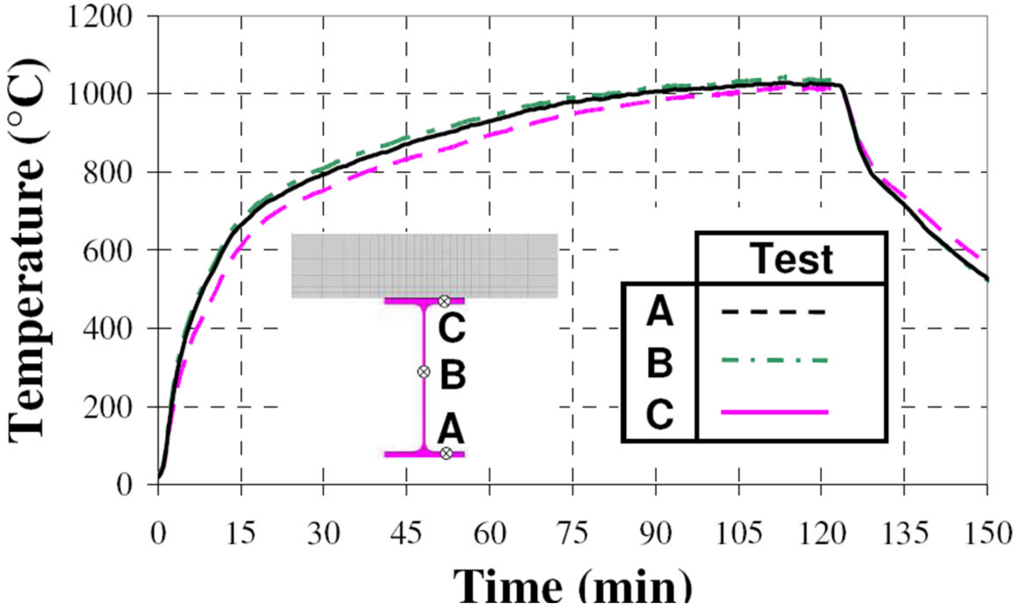
[7]

Reference case

FRACOF fire test - **Results**

Fire (gas) temperature ⟹





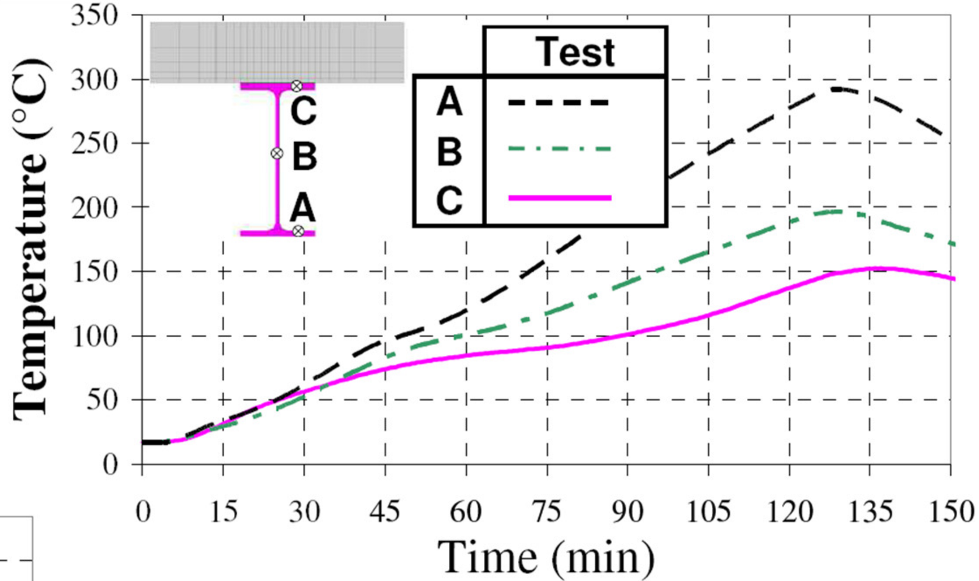
⟸ Heating of unprotected steel beams

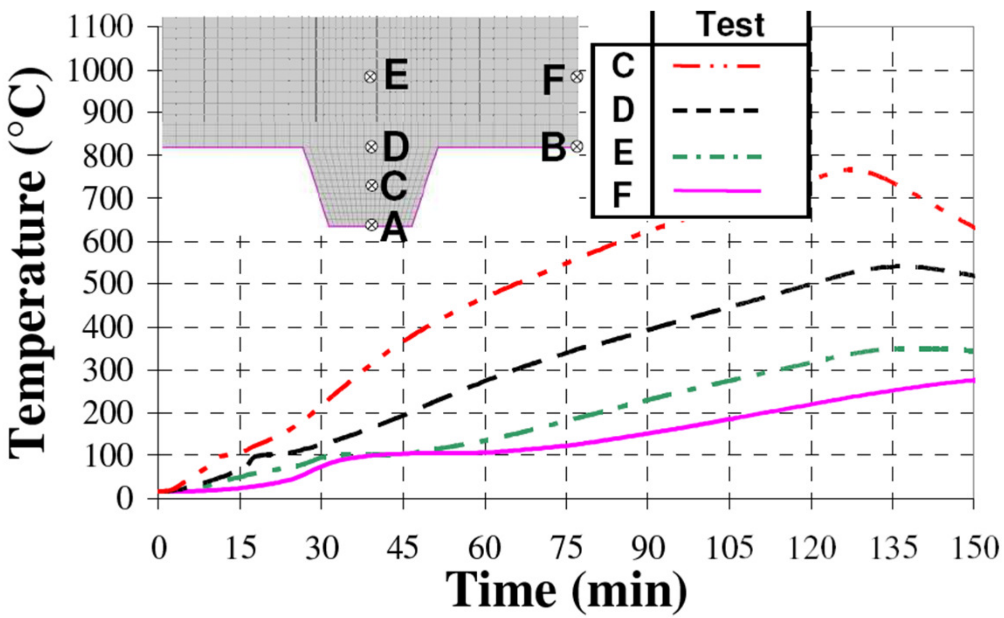
[7]

Reference case

FRACOF fire test - **Results**

Heating of protected steel ⟹





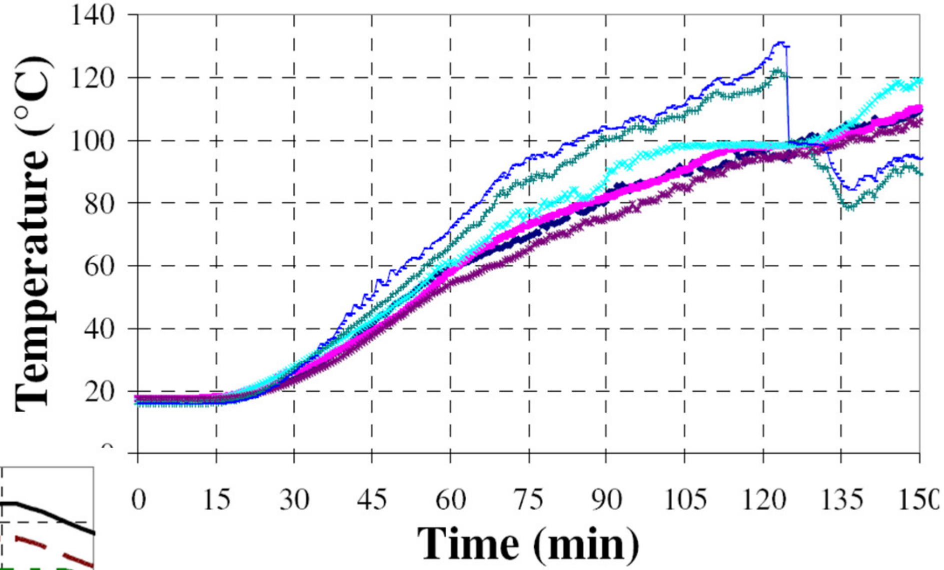
⟸ Heating of composite slab

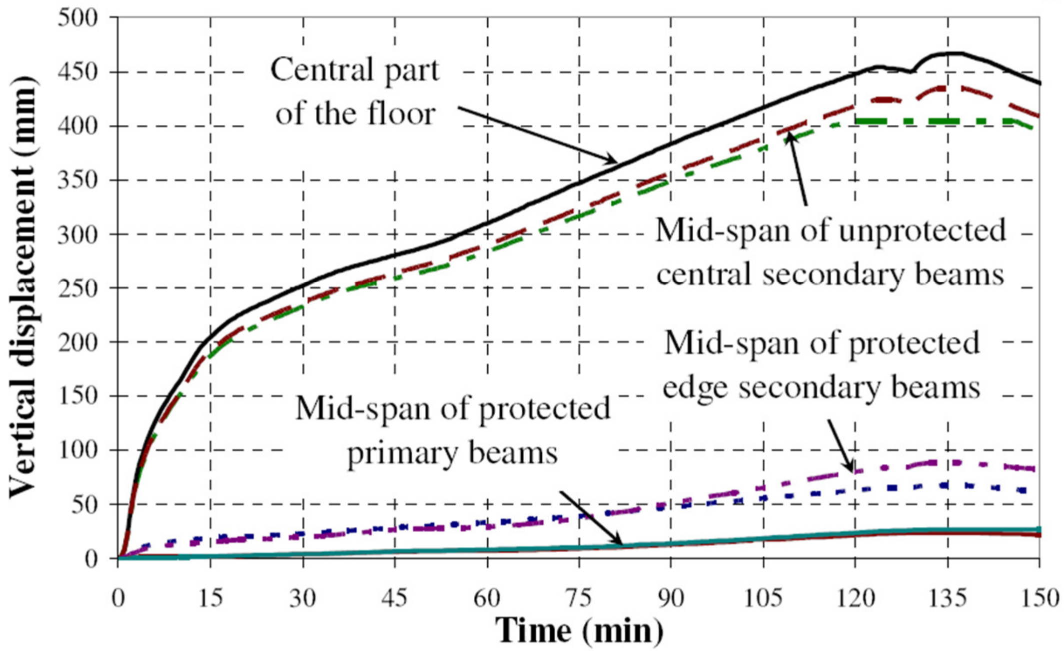
[7]

Reference case

FRACOF fire test - **Results**

Temperatures at the unexposed face of slab ⇨





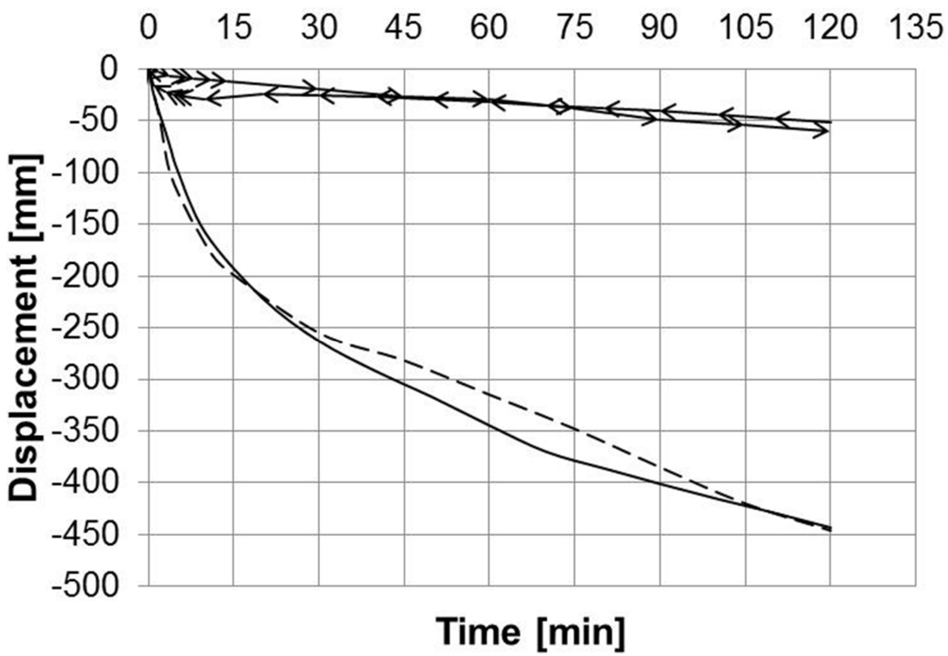
⇦ Deflection of the floor

[7]

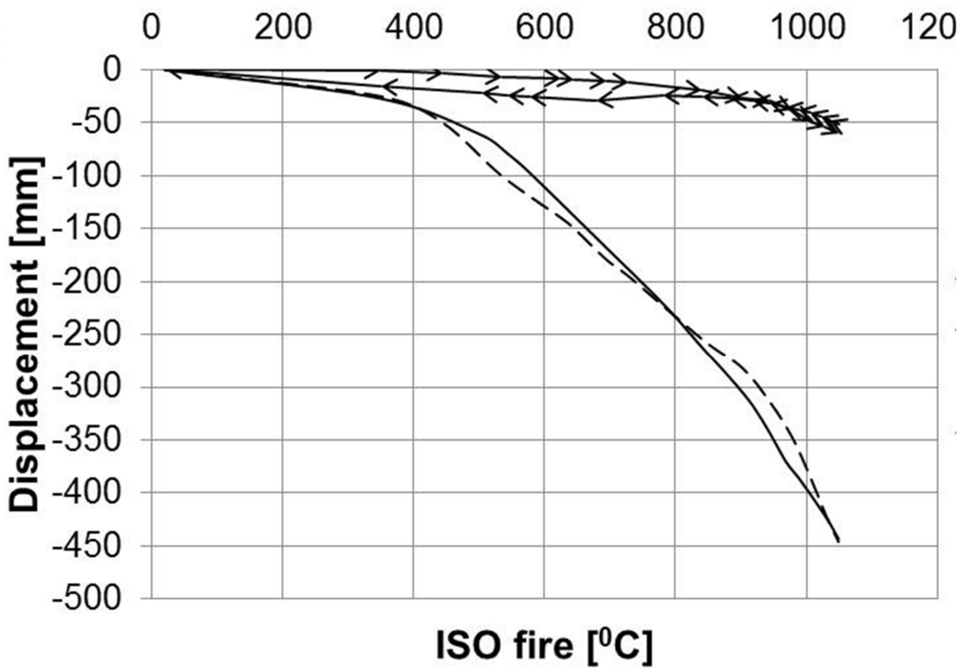
Validation

Displacement

with respect to time



with respect to temperature



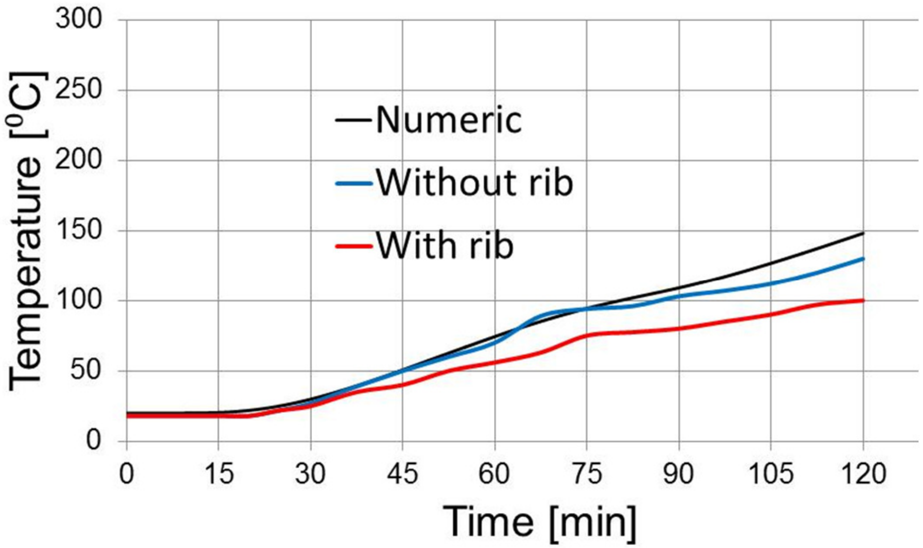
Legend:

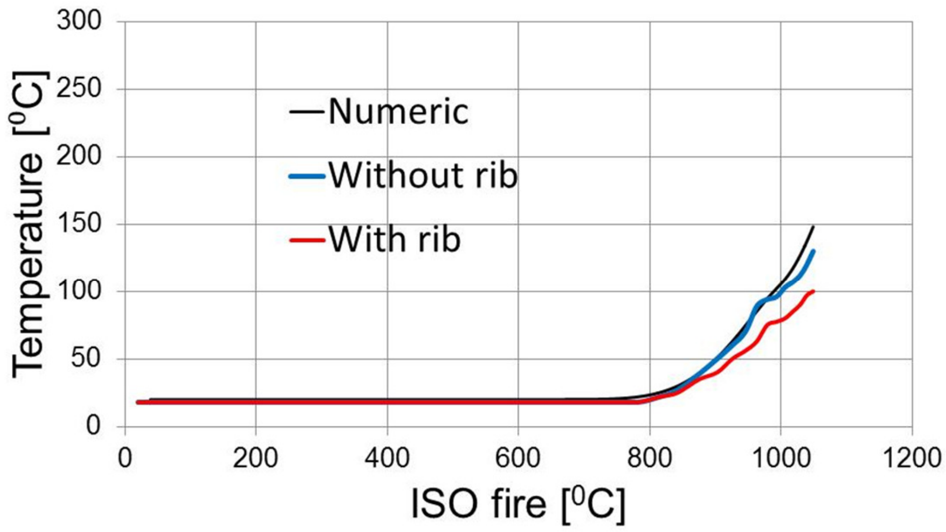
Central point-numeric
Central Point-test
Protected IPE300-numeric
Protected IPE300-test

Validation

Temperature

Unexposed side of the slab





with respect to time

with respect to temperature

Benchmark model

Materials properties

Consideration: only strength of materials is affected by temperature!!! (EN 1994-1-2)

Thermal analysis

Material	Density [kg/m^3]	Conductivity [W/m K]	Specific heat [J/kg K]
Steel	7850	40	550
Concrete	2400	0.9	1050

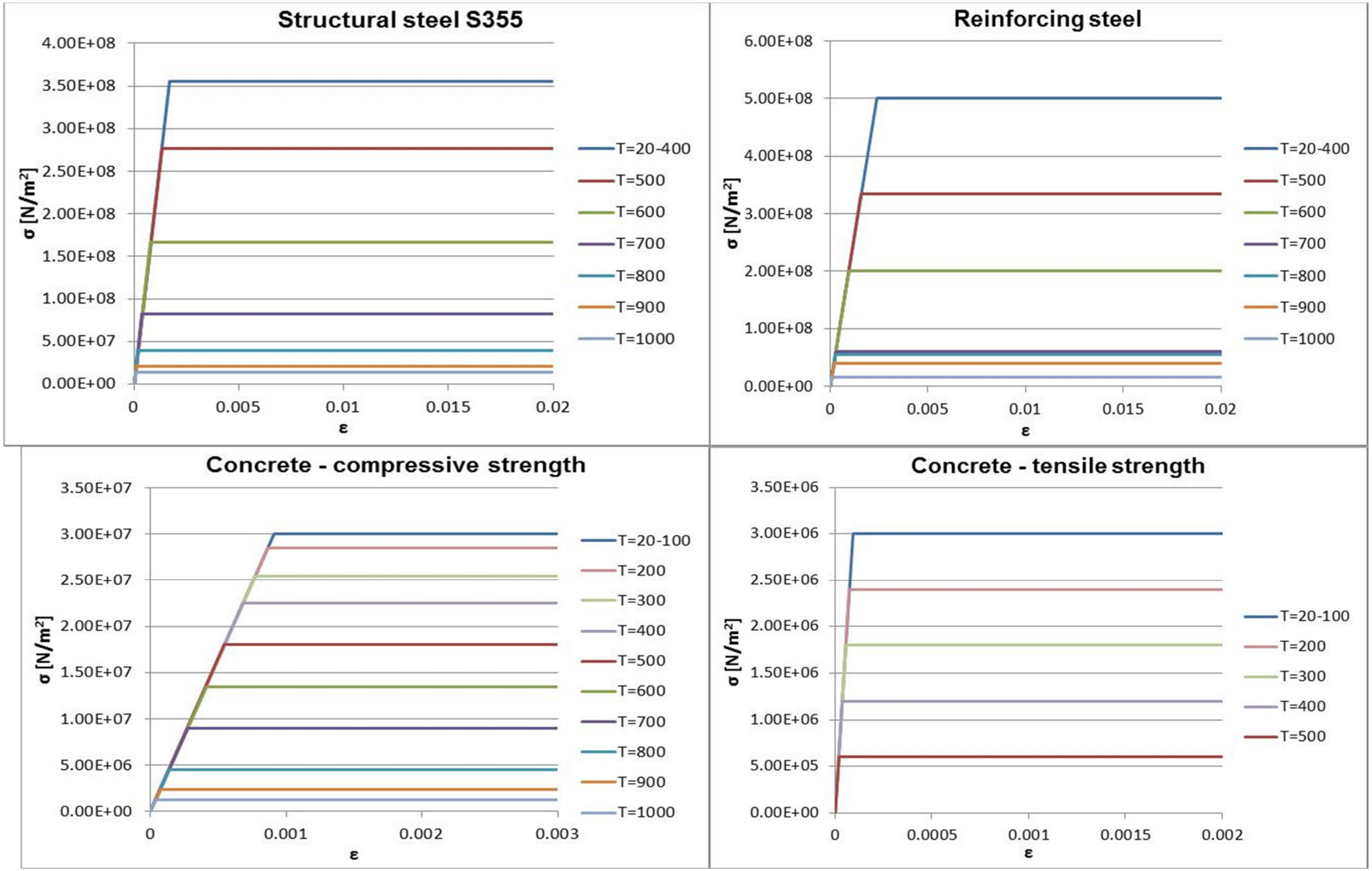
Mechanical ananlysis

Steel	E [N/m^2]	ν	σ_y [N/m^2]	α [1/C]
S235	2.1e11	0.3	235.0e6	1.4e-5
S355	2.1e11	0.3	355.0e6	1.4e-5
S500	2.1e11	0.3	500.0e6	1.4e-5

Concrete	E [N/m^2]	ν	f_c [N/m^2]	f_t [N/m^2]	α [1/C]
C30/37	3.3e10	0.2	30.0e6	3.0e6	1.0e-5

Benchmark model

Materials properties

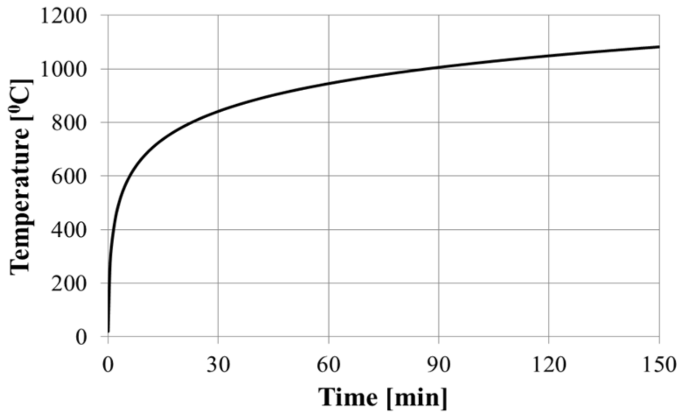


Benchmark model

Loads

Thermal load:

- constant temperature for unprotected beams,
- gradients through protected beams section,
- imported temperature field for slab



Standard fire curve

Mechanical load:

- sand bags: 3870 N/m^2
- selfweight: 3280 N/m^2

7150 N/m^2 – uniform pressure on the slab

Numerical model - Basics

Heat transfer analysis in Abaqus summary

- uncoupled heat transfer analysis
- sequentially coupled thermal-stress analysis
- fully coupled thermal-stress analysis,
- fully coupled thermal-electric-structural analysis,
- adiabatic analysis,
- coupled thermal-electrical analysis
- cavity radiation

Analysis

- Steady-state
- Transient

[4]

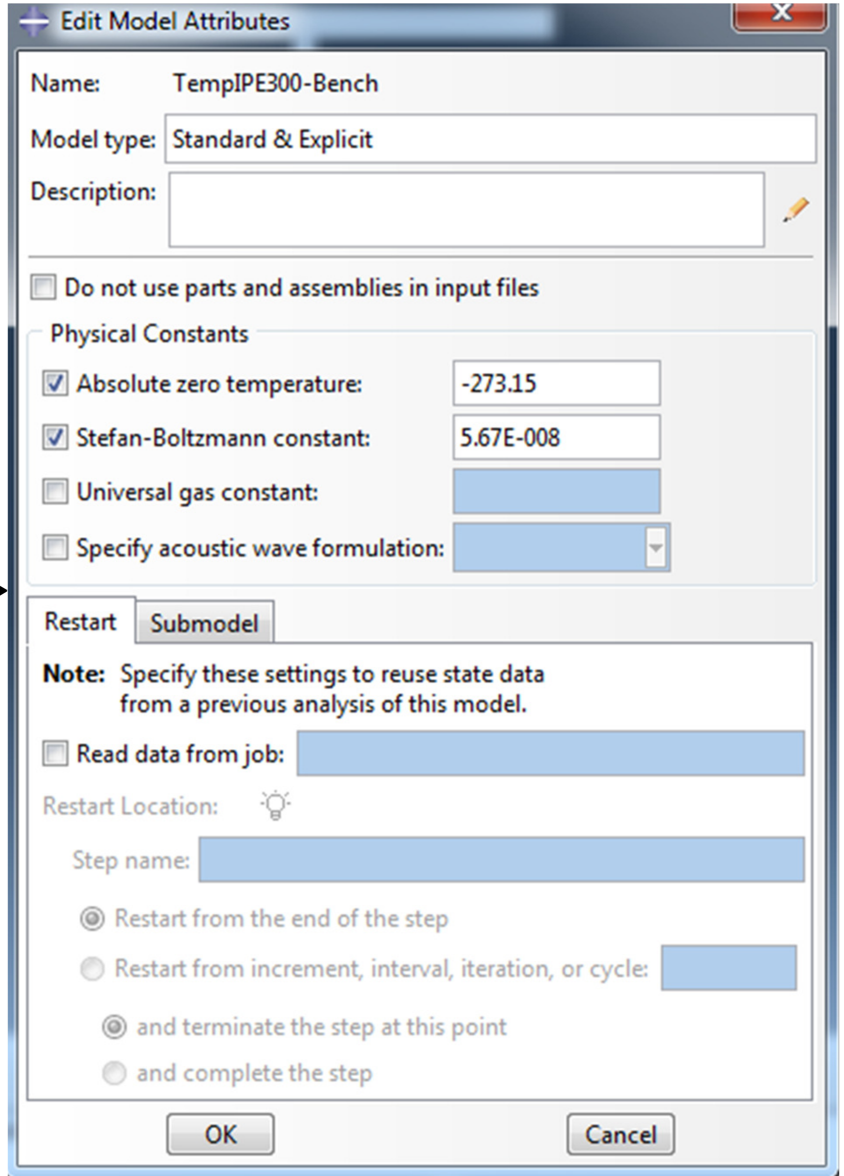
Numerical model - Basics

Settings

ABAQUS has no settings for units system

Measurement units are chosen by the user and should be consistent throughout all model(s)

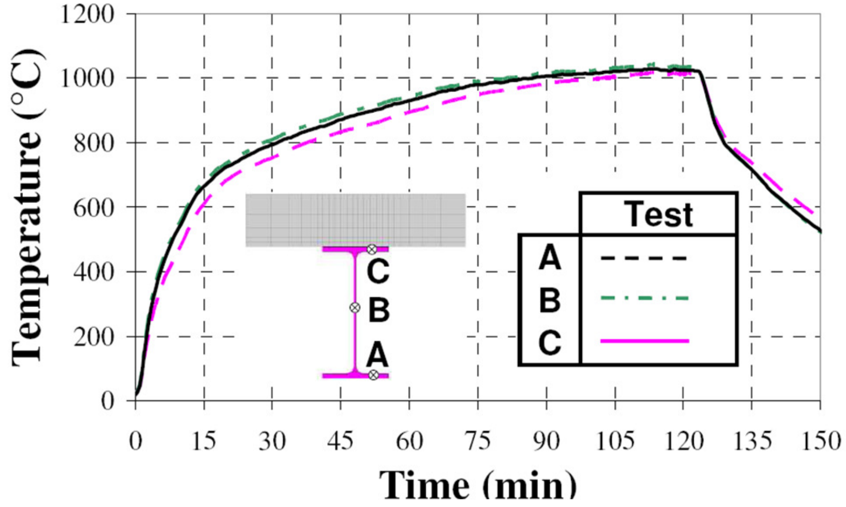
For the benchmark the units are:
N, m, s, ^{0}C



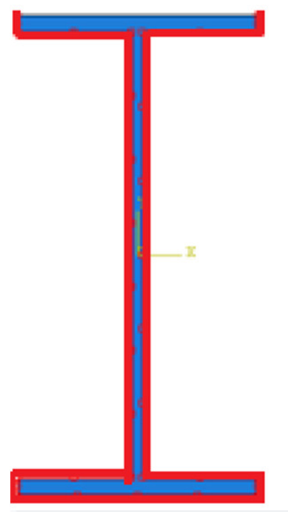
Numerical model

Temperature field for secondary, unprotected beams

From test➔ uniform temperature



Create a 2D heat transfer model for the IPE300 section



Define interactions to the environment:

- convection
- radiation

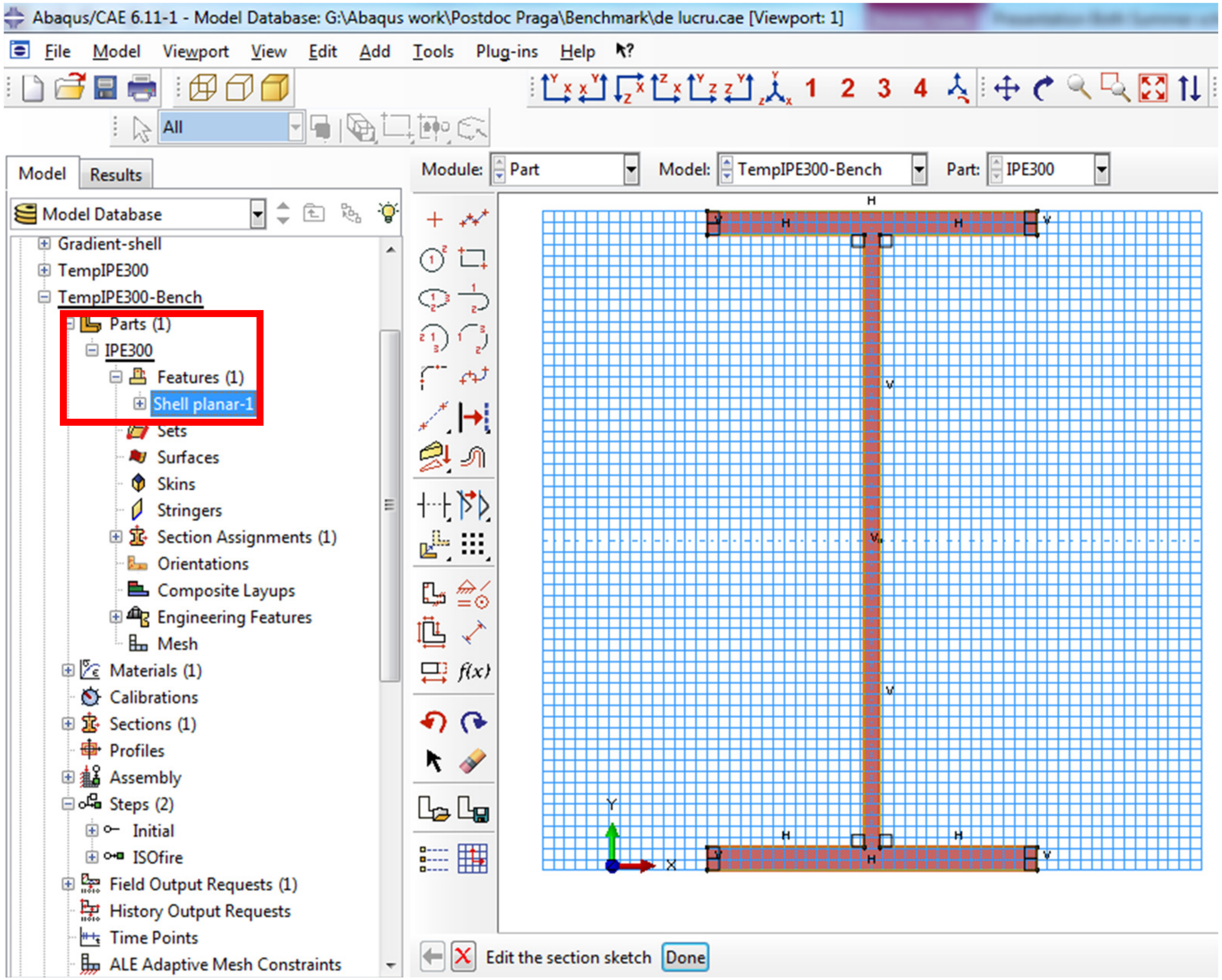
Obtain temperature field which will be used in the composite slab model

Numerical model

Temperature field for secondary, unprotected beams

Create part:

2D shell planar

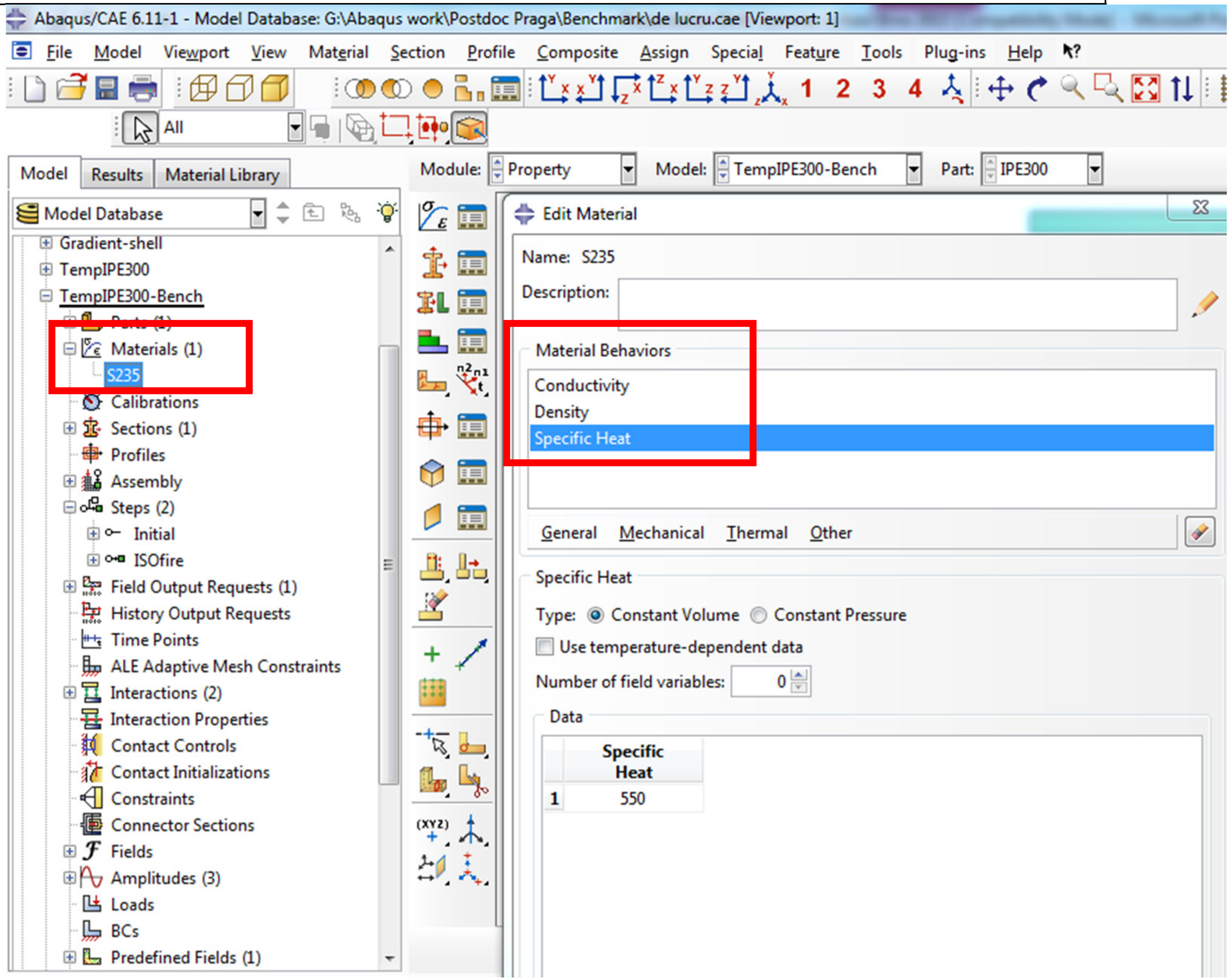


Numerical model

Temperature field for secondary, unprotected beams

Define material:

- conductivity
- specific heat
- density

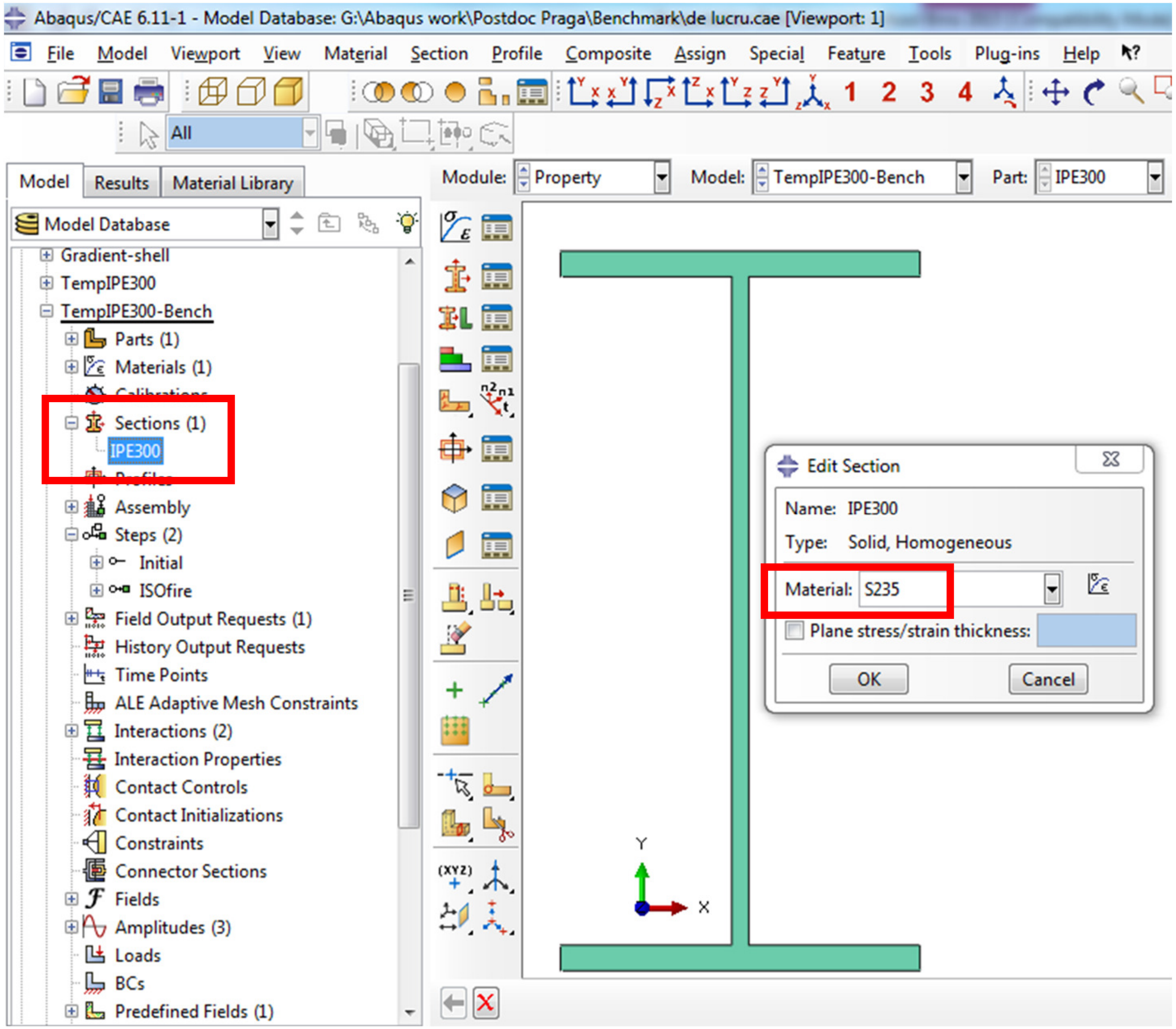


Numerical model

Temperature field for secondary, unprotected beams

Define section property:

- Solid homogeneous

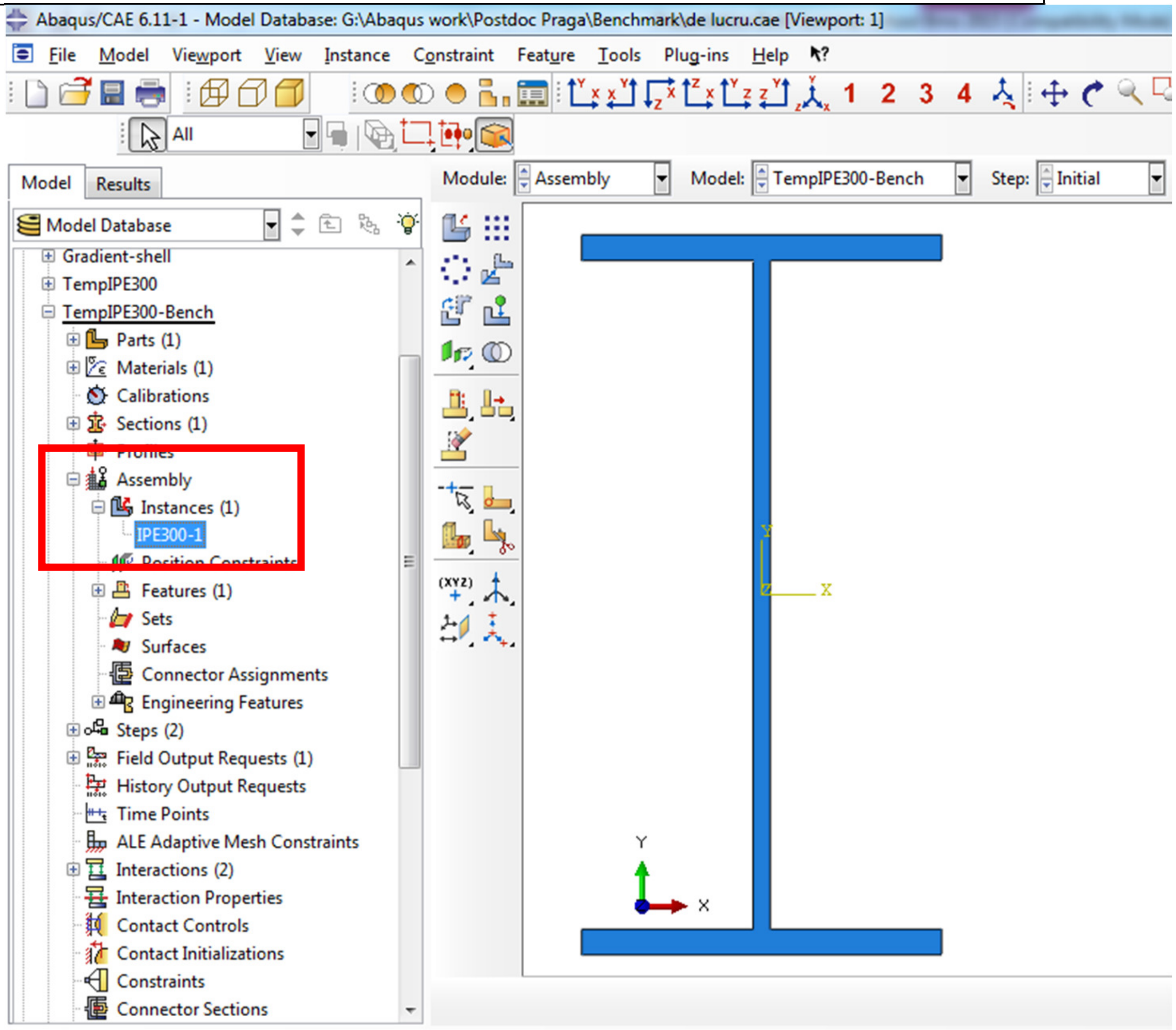


Numerical model

Temperature field for secondary, unprotected beams

Create instances:

- IPE300

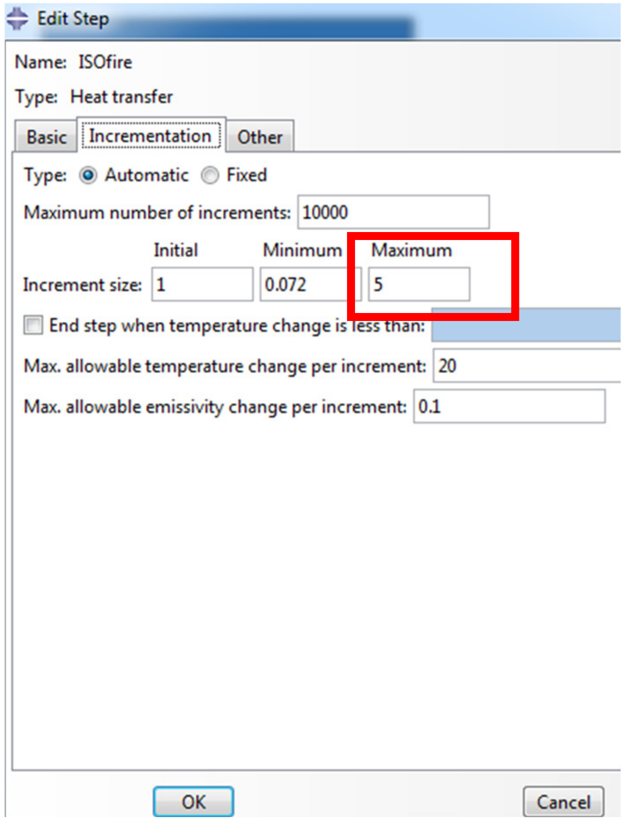


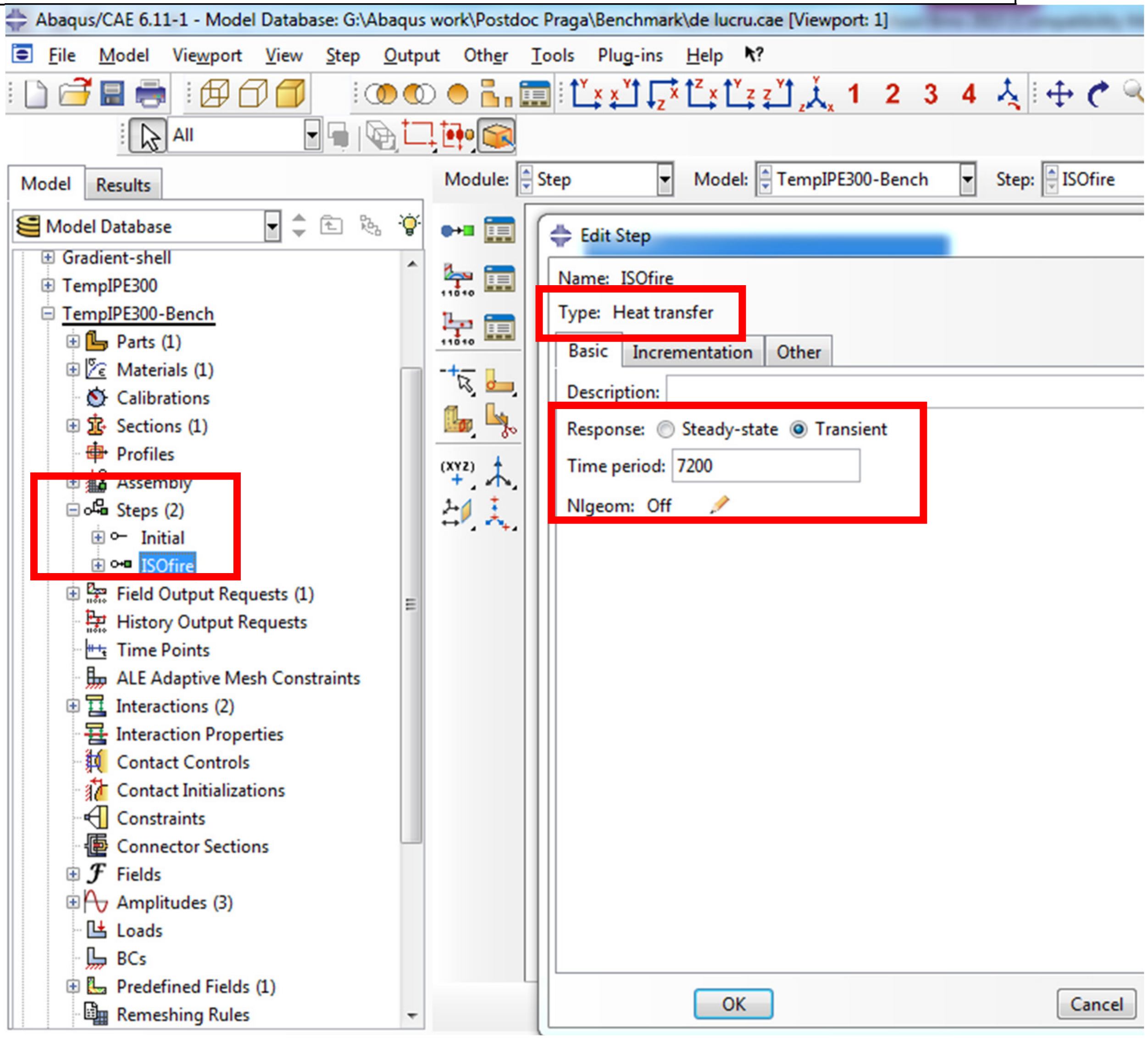
Numerical model

Temperature field for secondary, unprotected beams

Define steps:

- Heat transfer



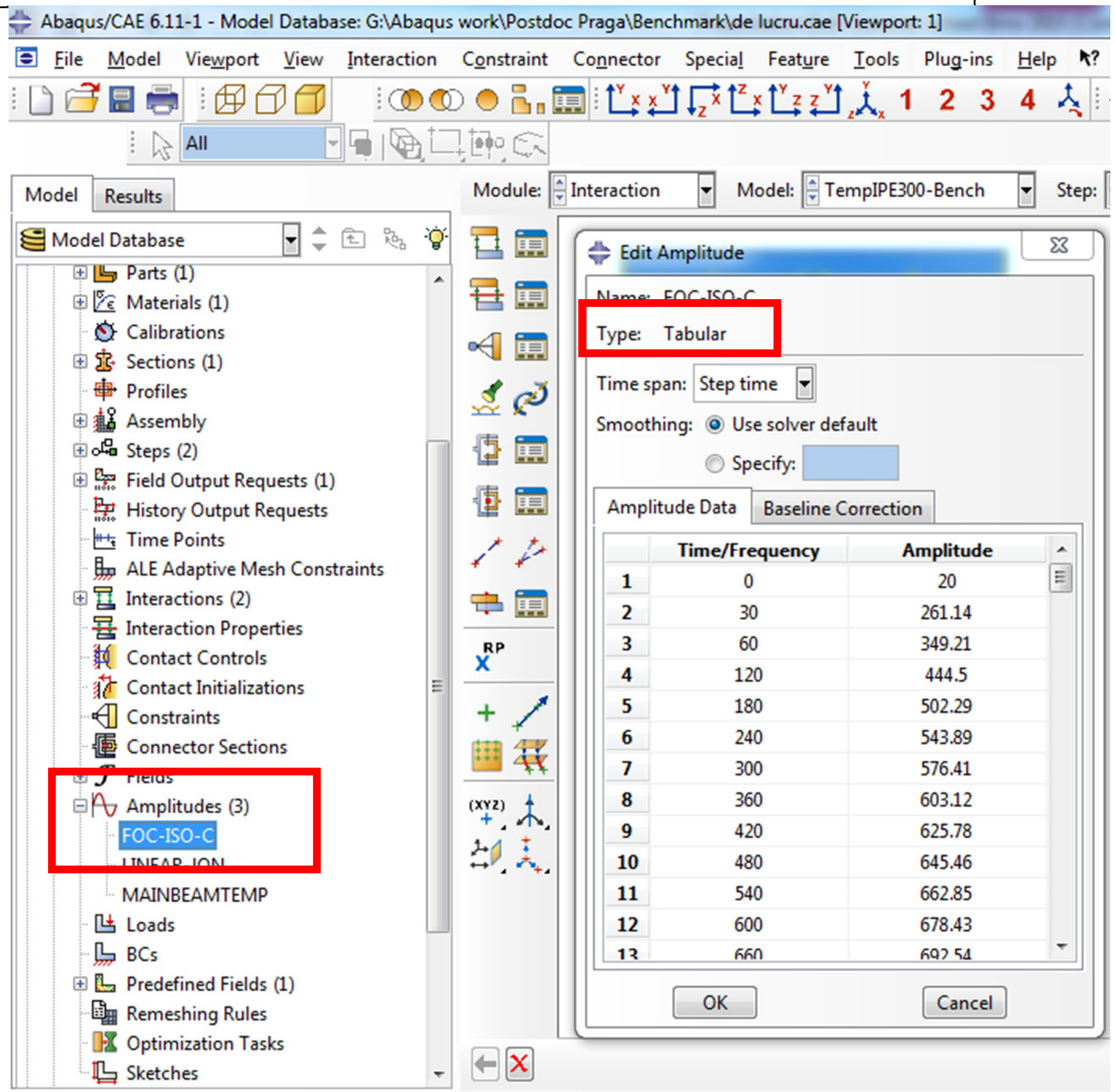


Numerical model

Temperature field for secondary, unprotected beams

Define fire curve:

- Amplitude

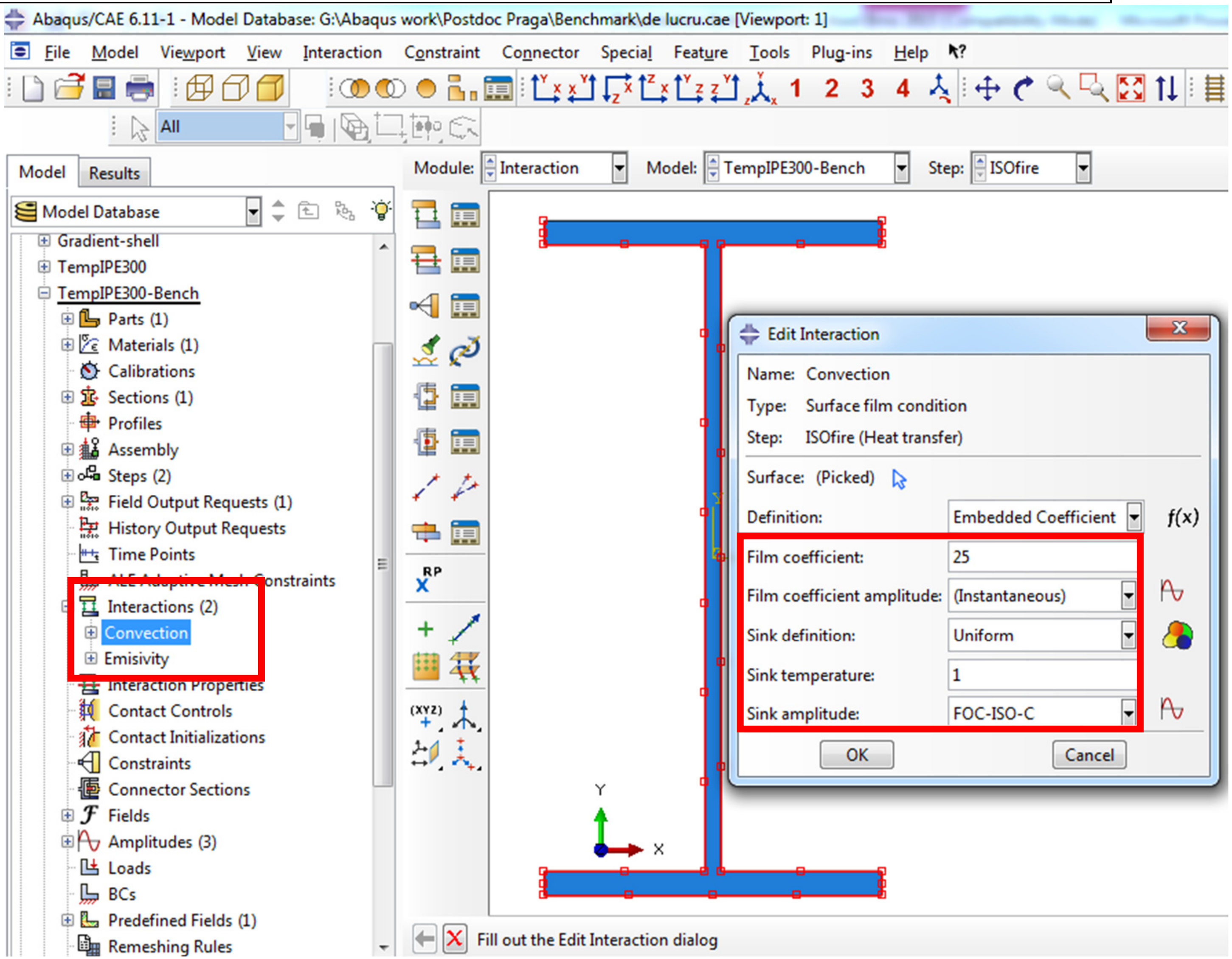


Numerical model

Temperature field for secondary, unprotected beams

Define interactions:

- Convection (surface film condition)

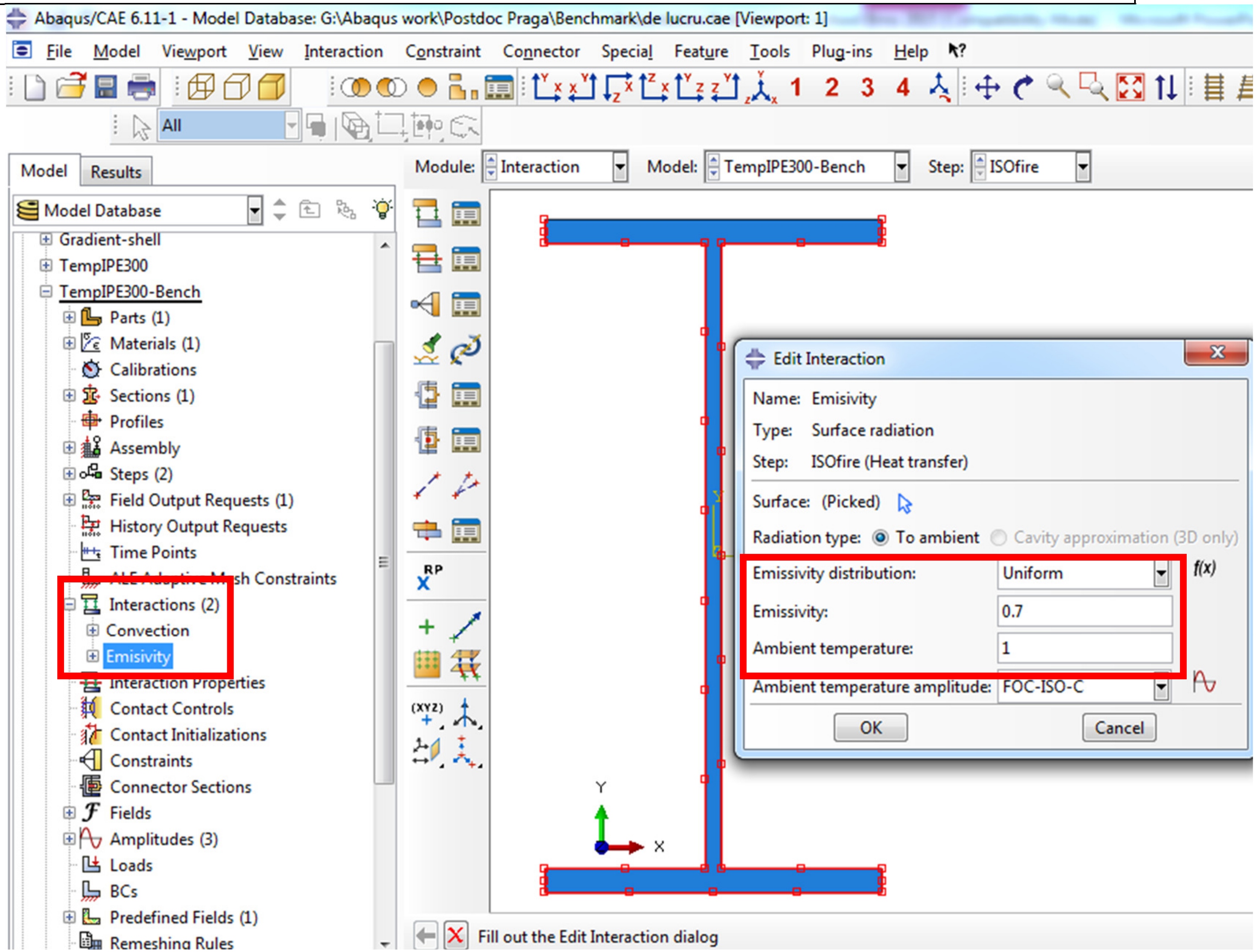


Numerical model

Temperature field for secondary, unprotected beams

Define interactions:

- Radiation (surface radiation)

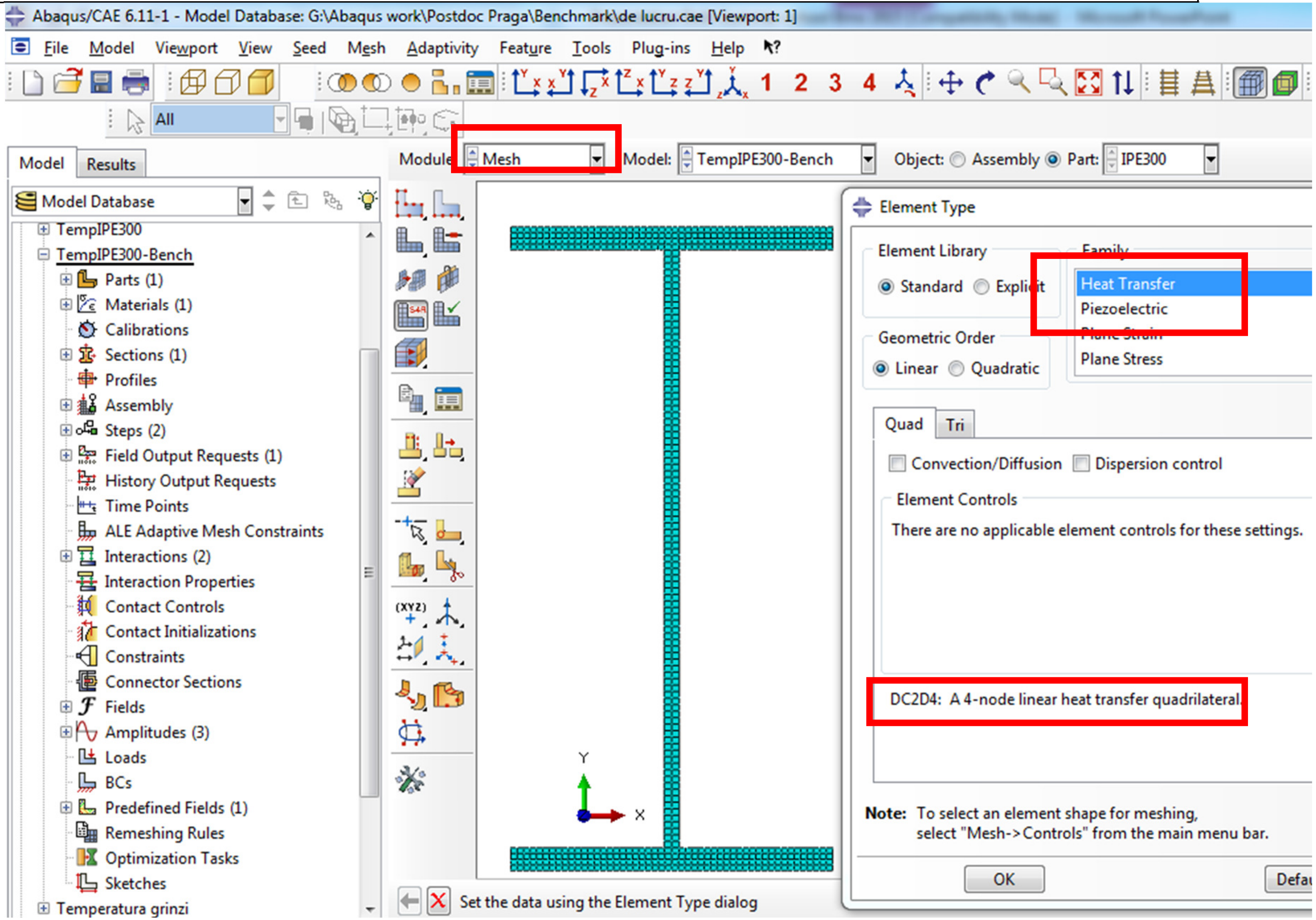


Numerical model

Temperature field for secondary, unprotected beams

Define mesh:

-DC2D4

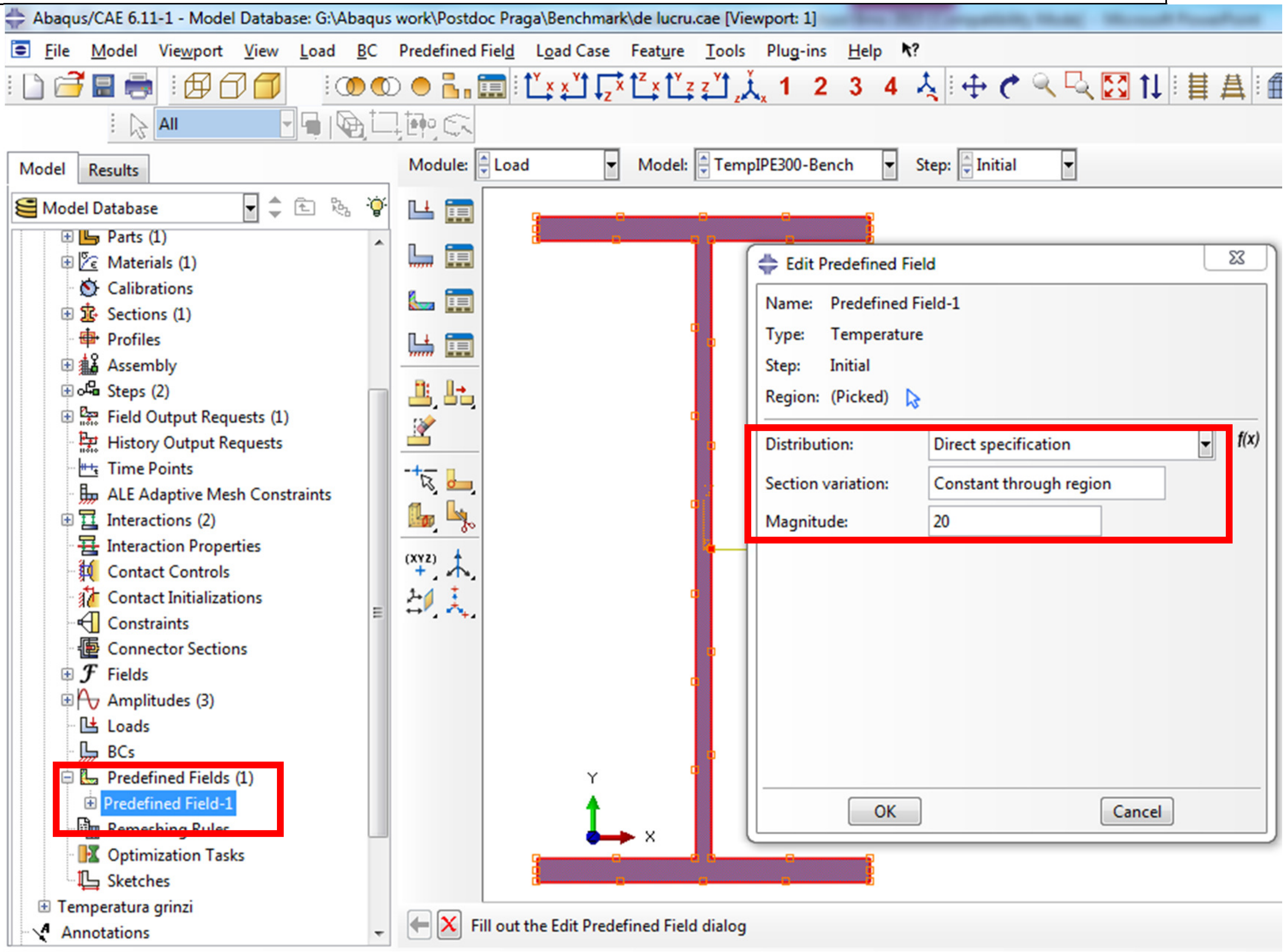


Numerical model

Temperature field for secondary, unprotected beams

Define initial temperature:

- Predefined field

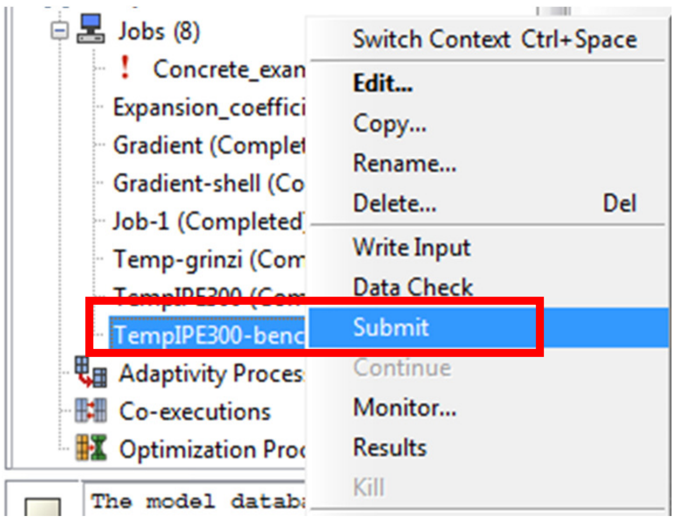


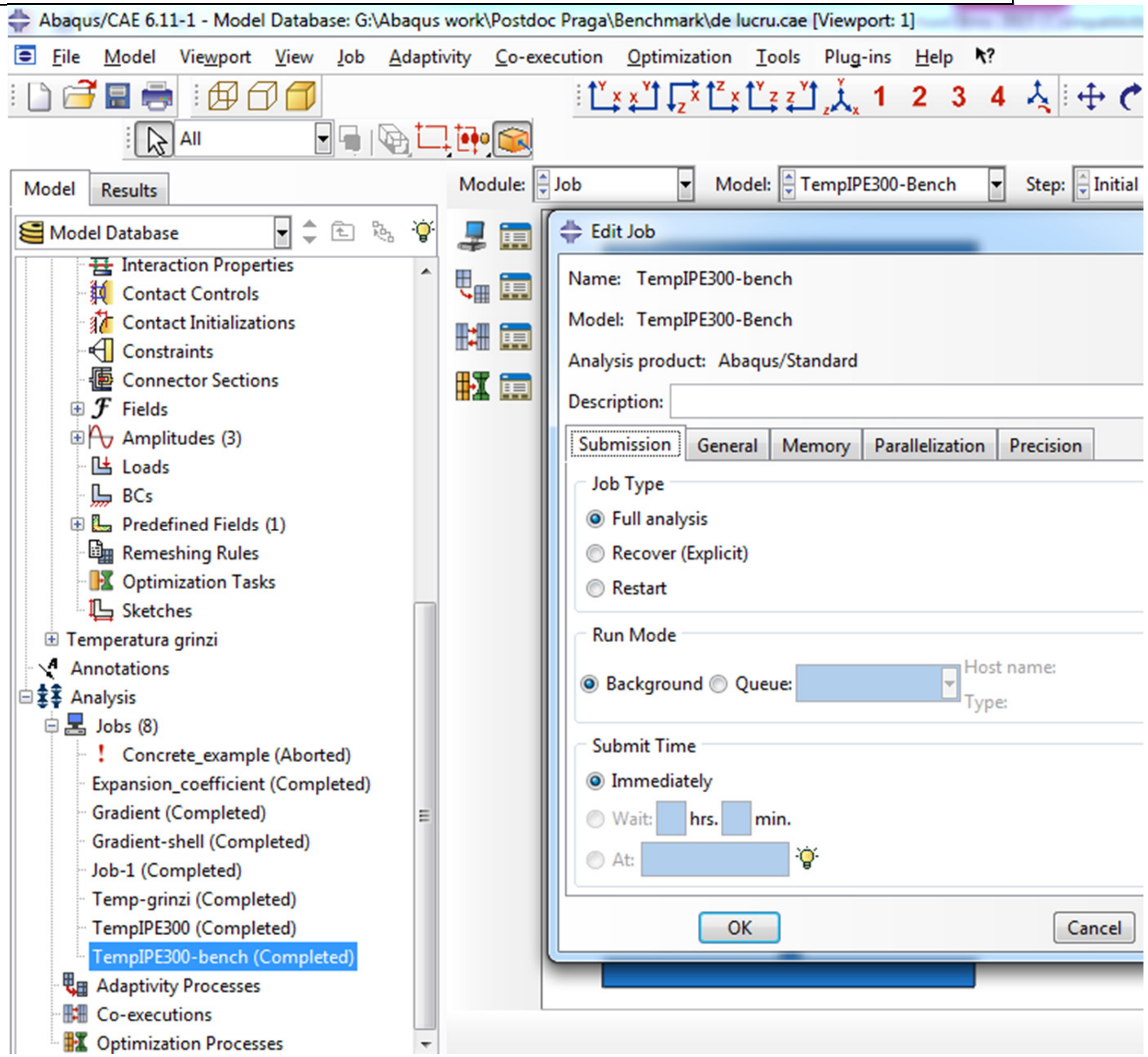
Numerical model

Temperature field for secondary, unprotected beams

Create job and run analysis:

- Jobs
- Submit



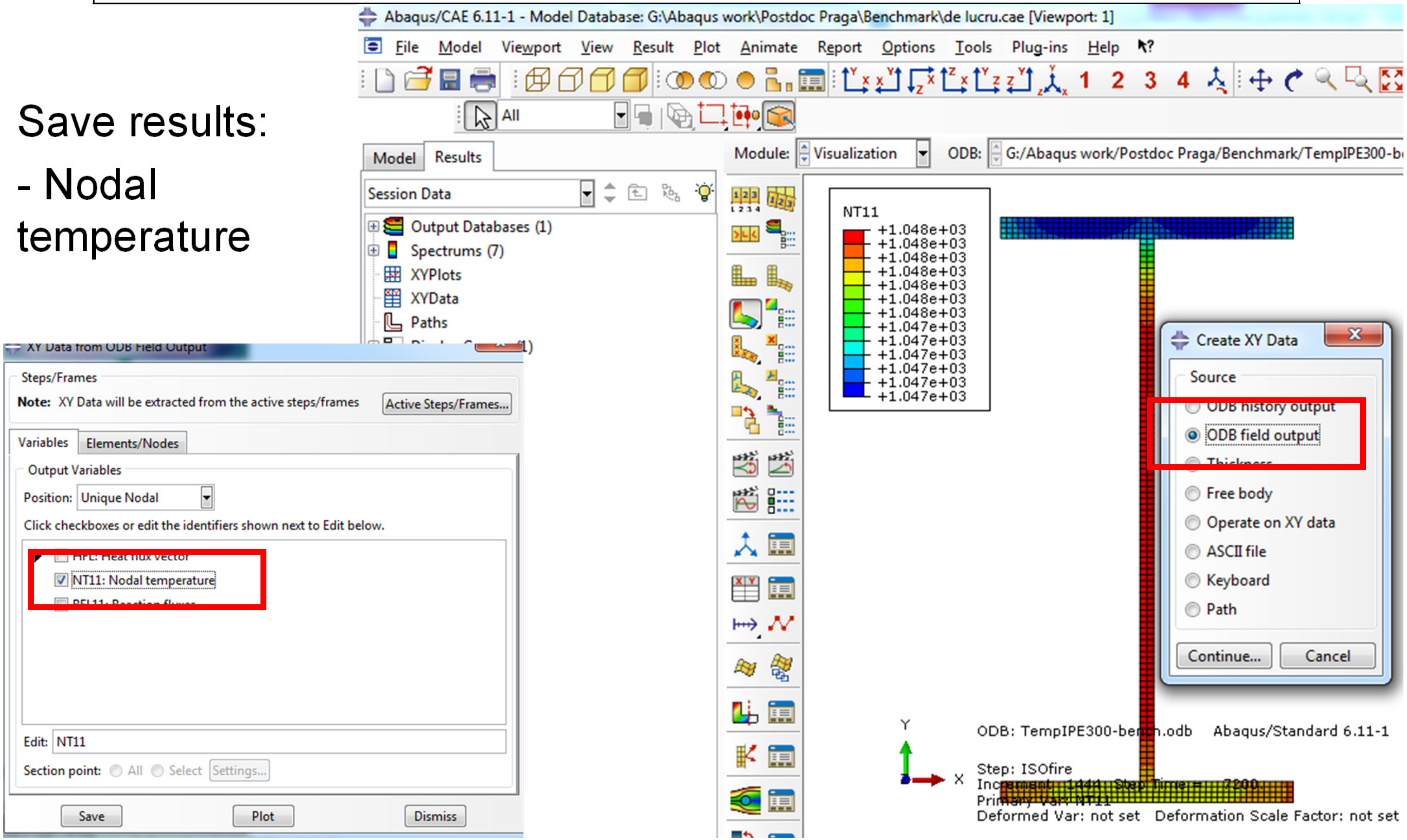


Numerical model

Temperature field for secondary, unprotected beams

Save results:

- Nodal temperature

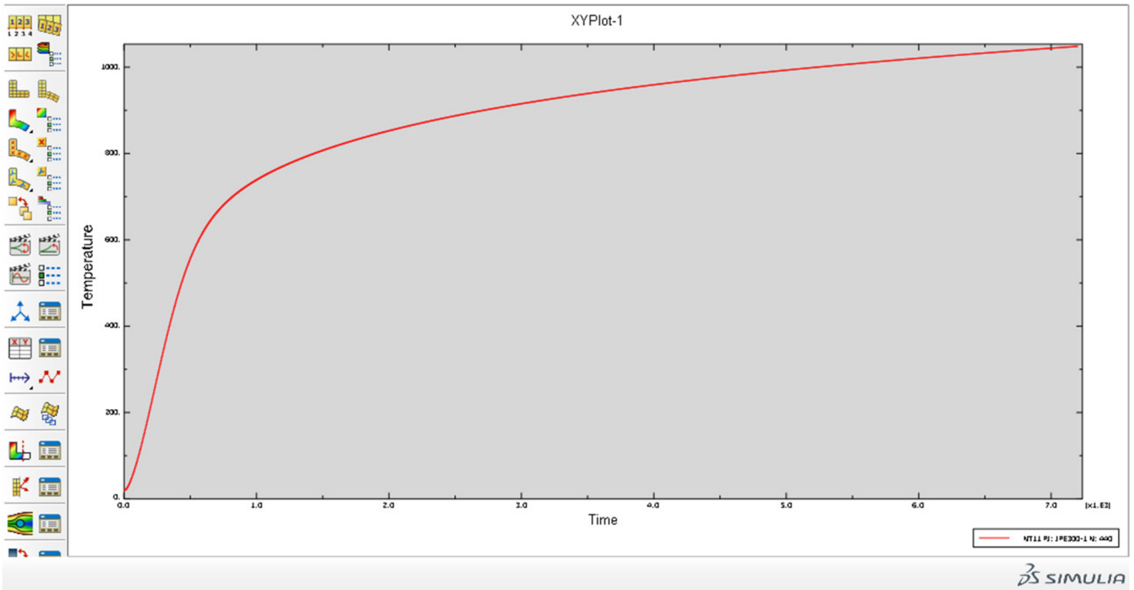


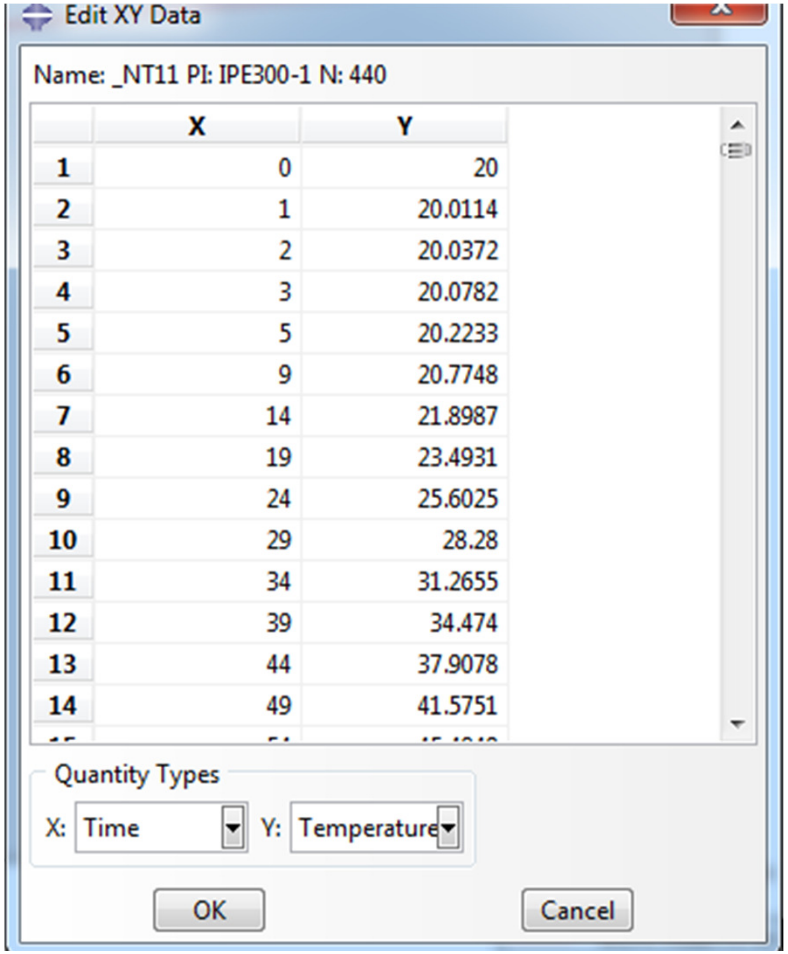
Numerical model

Temperature field for secondary, unprotected beams

Save results:

- Nodal temperature





Edit XY Data

Name: _NT11 PI: IPE300-1 N: 440

	X	Y
1	0	20
2	1	20.0114
3	2	20.0372
4	3	20.0782
5	5	20.2233
6	9	20.7748
7	14	21.8987
8	19	23.4931
9	24	25.6025
10	29	28.28
11	34	31.2655
12	39	34.474
13	44	37.9078
14	49	41.5751

Quantity Types

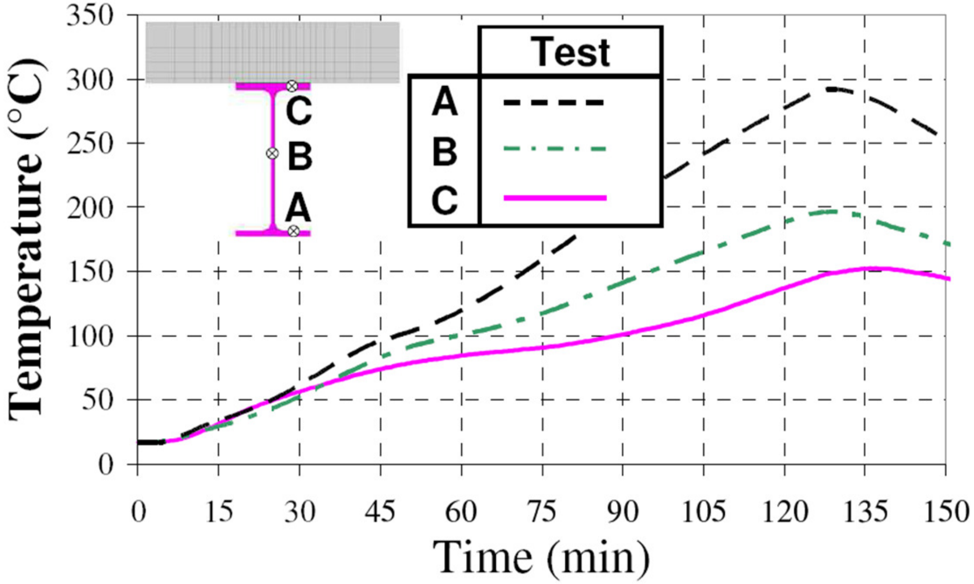
X: Time Y: Temperature

OK Cancel

Numerical model

Temperature field for protected beams

Temperature from test: difference between top and bottom flange

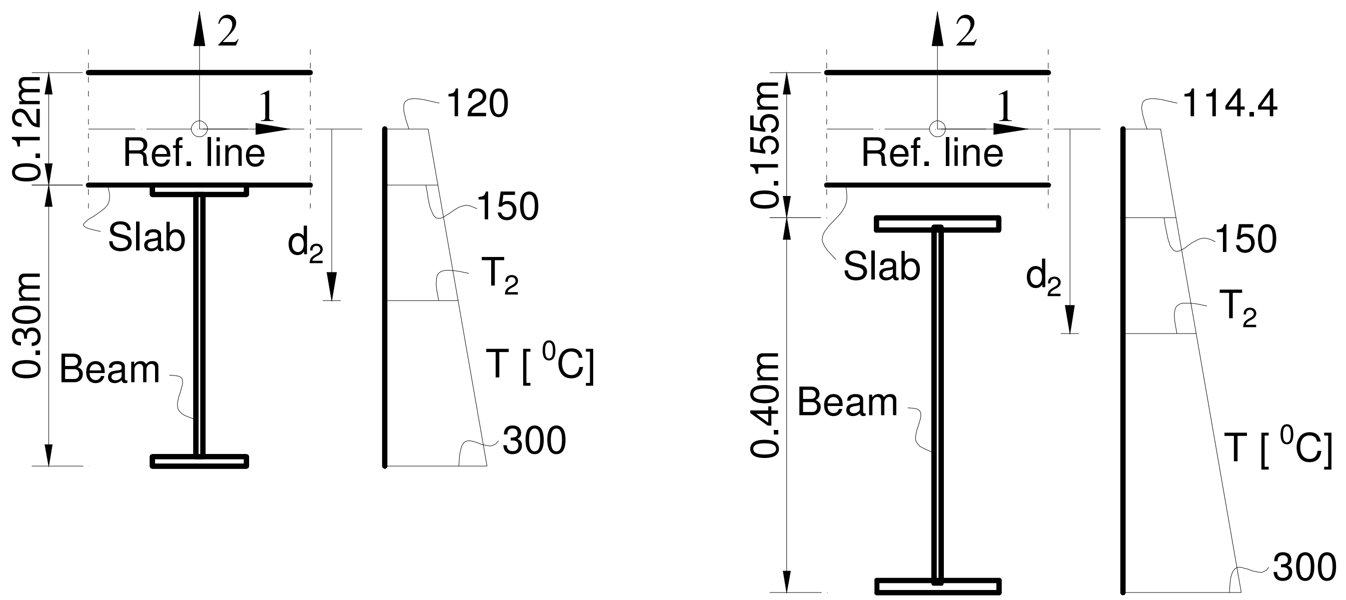


It is defined as a *predefined field* gradient through beam section in the composite slab model (no need for an additional model)

Numerical model

Temperature field for protected beams

Beam elements sections are defined function of a reference line

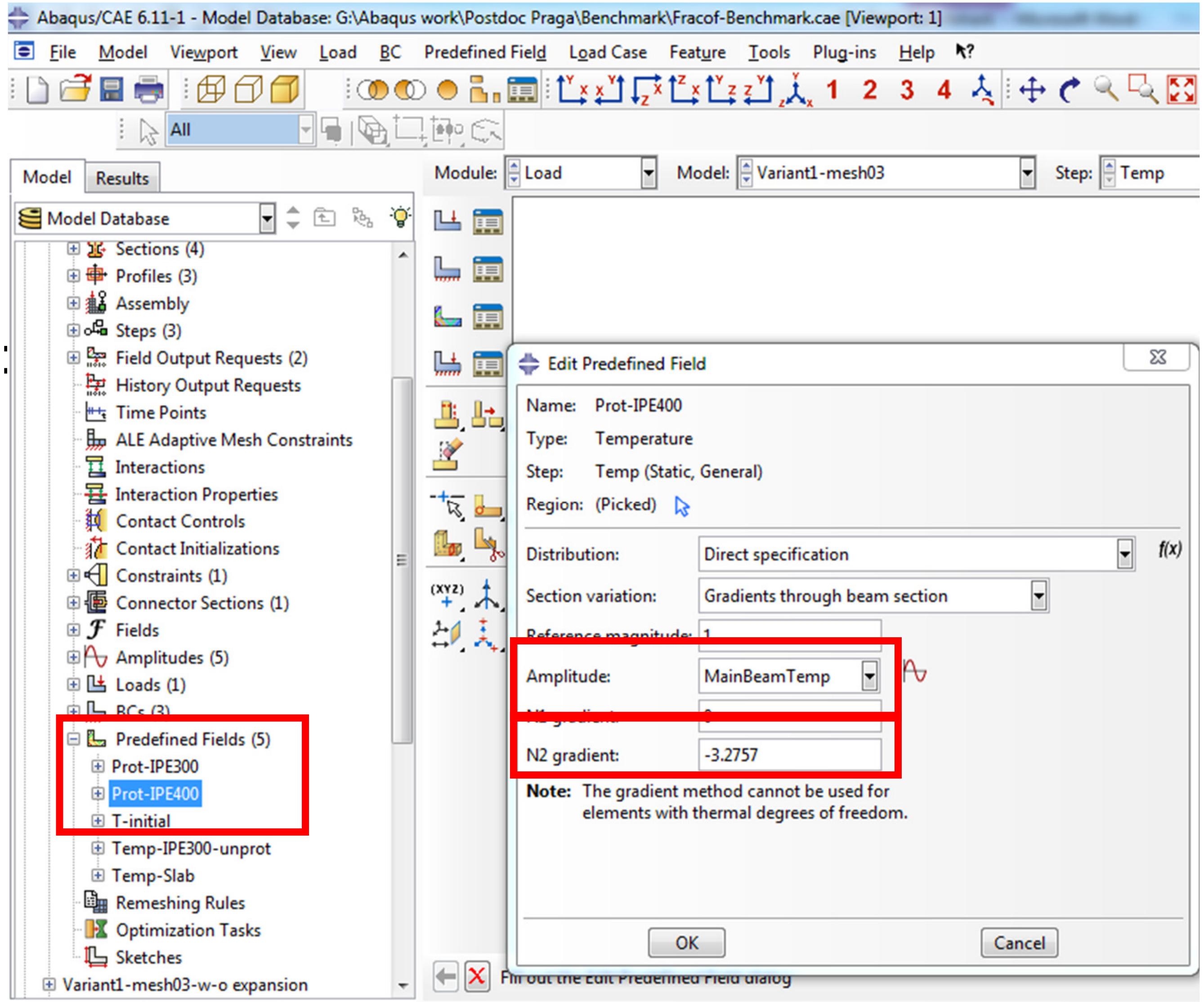


Numerical model

Temperature field for protected beams

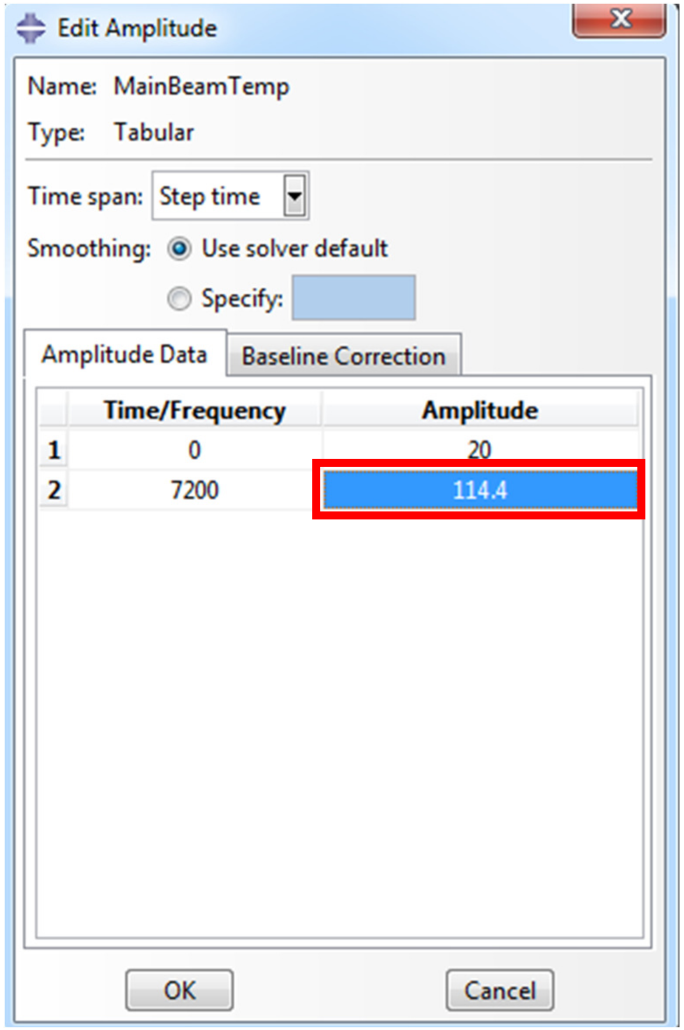
Input for *predefined field* of gradient through beam section:

- amplitude
- gradient

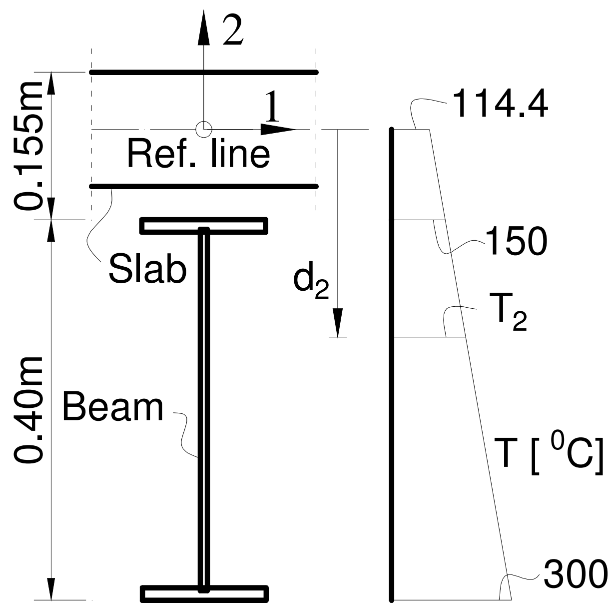


Numerical model

Temperature field for protected beams



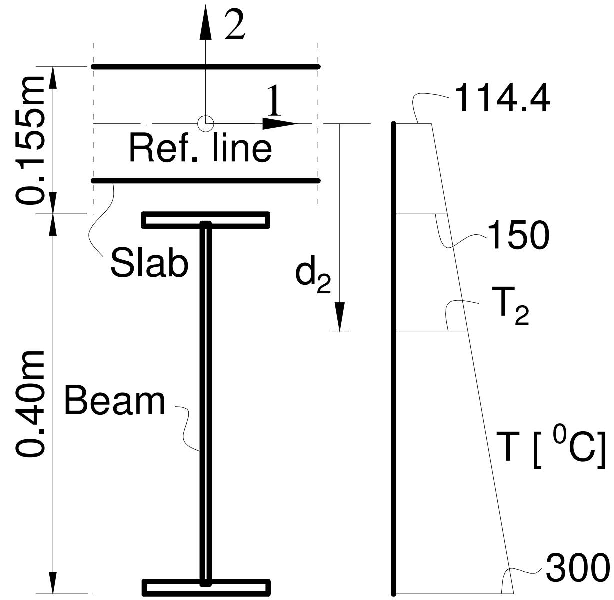
Amplitude is function of reference line temperature obtained by linear interpolation



Numerical model

Temperature field for protected beams

Determination of gradient



$$\theta_{ref}\,(1 + d_2 \cdot x) = \theta_2$$

x – *gradient*

θ_{ref} – temperature at reference line level,

d_2 – distance from reference line to a point along direction 2;

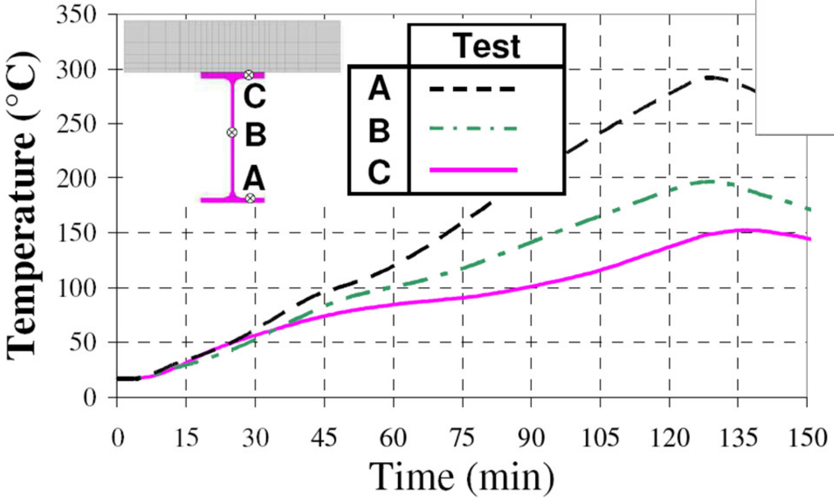
θ_2 – temperature at distance d_2

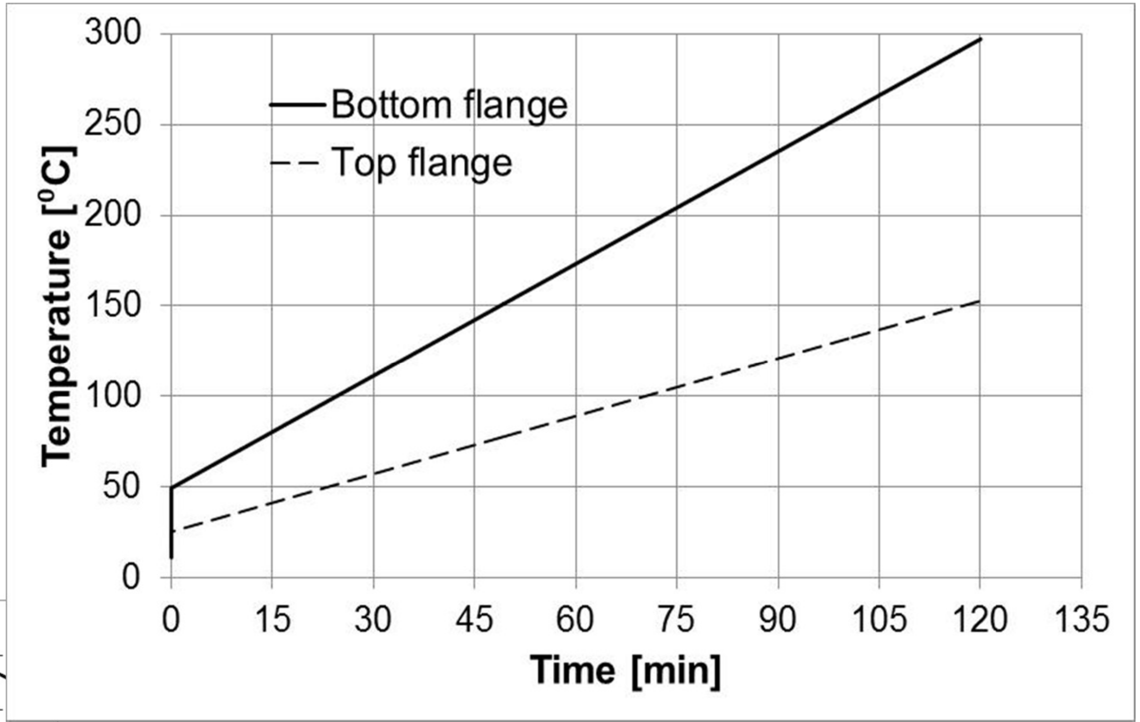
x ➔ -3.2757

Numerical model

Temperature field for protected beams

Results



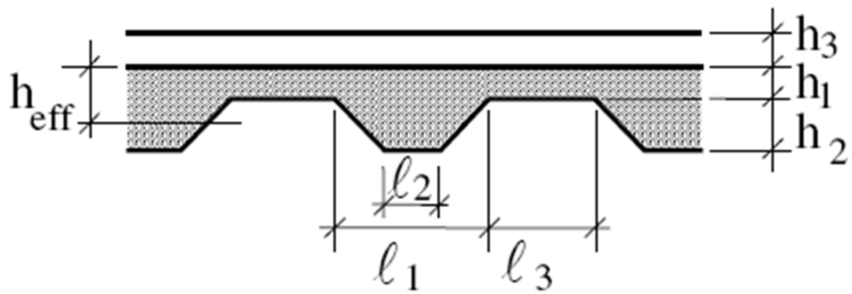


Numerical model

Temperature field for concrete slab

Sequentially coupled thermal-displacement analysis ➔ Create a separate heat transfer model (initial model for mechanical analysis – similar coordinates of slab)

It is considered an equivalent thickness of slab according to EN1994-1-2 Annex D



$$h_{eff} = h_1 + 0{,}5\, h_2 \left(\frac{\ell_1 + \ell_2}{\ell_1 + \ell_3} \right) \qquad \text{for } h_2/h_1 \le 1{,}5 \text{ and } h_1 > 40 \text{ mm}$$

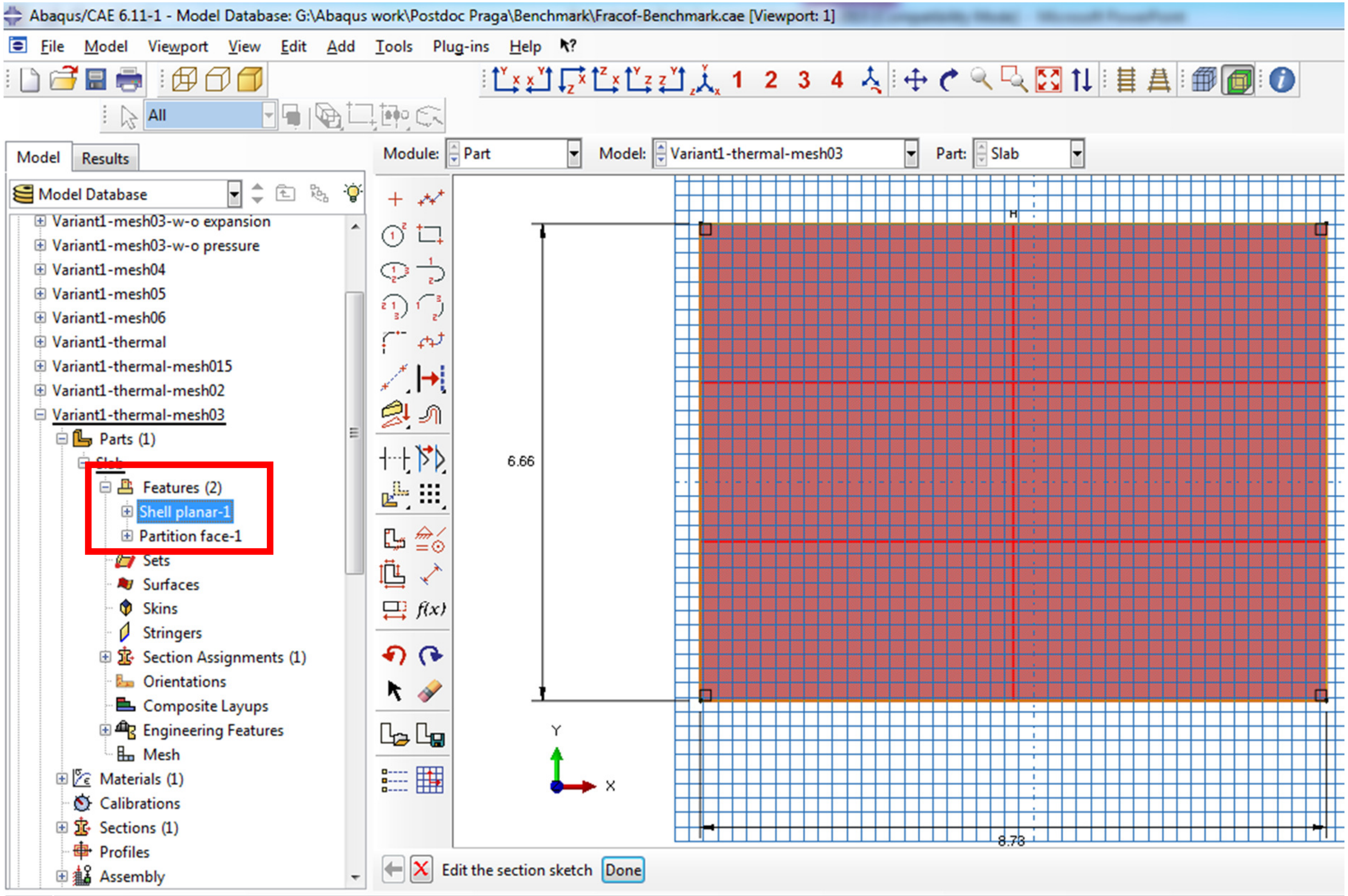
$$h_{eff} = h_1 \left[1 + 0{,}75 \left(\frac{\ell_1 + \ell_2}{\ell_1 + \ell_3} \right) \right] \qquad \text{for } h_2/h_1 > 1{,}5 \text{ and } h_1 > 40 \text{ mm}$$

Numerical model

Temperature field for concrete slab

Create the part and partition:

-3D shell planar



Numerical model

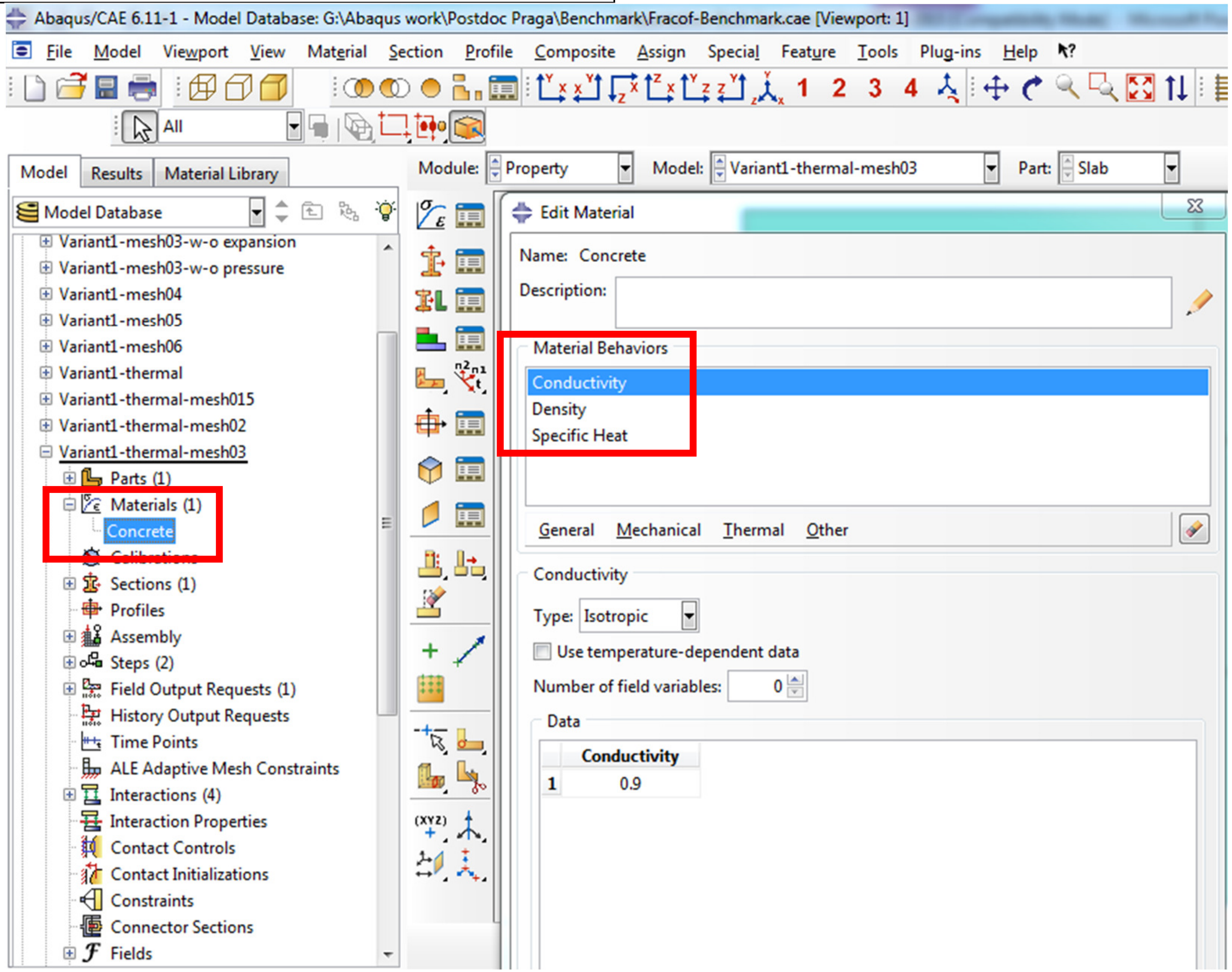
Temperature field for concrete slab

Define material:

-conductivity

-specific heat

-density



Numerical model

Temperature field for concrete slab

Define section:

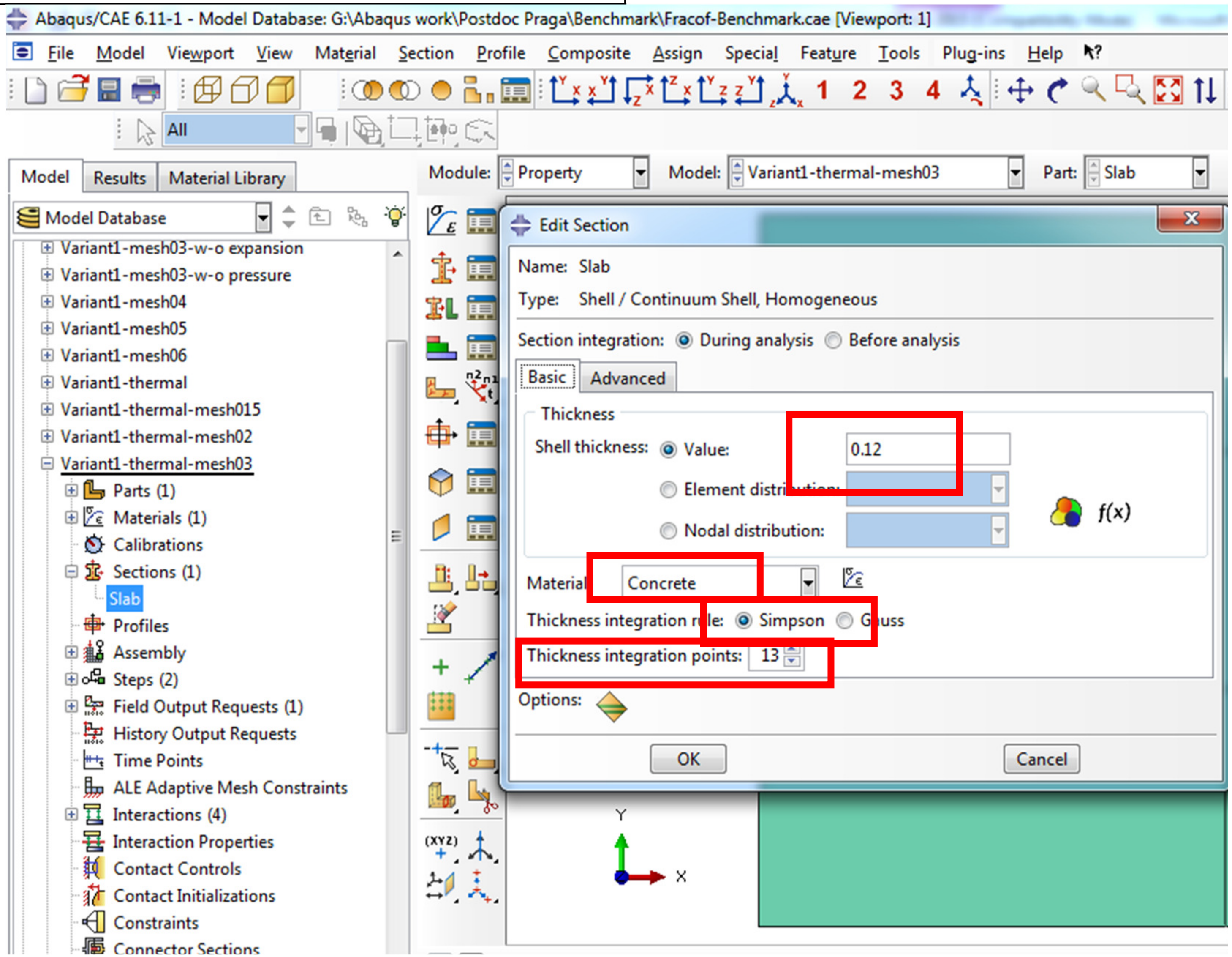
-thickness

-material

-integration rule

-integration points

(without reinforment, yet)

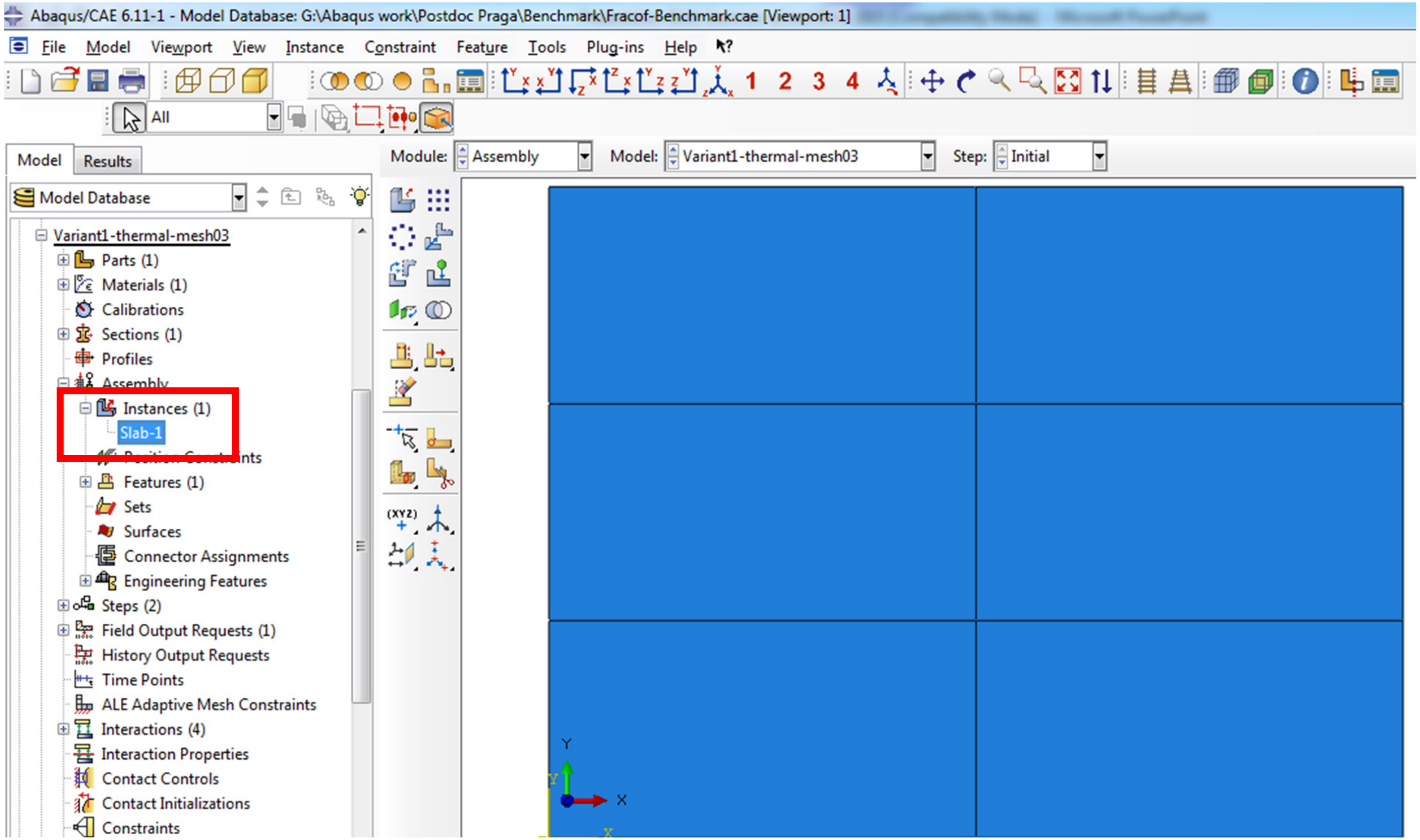


Numerical model

Temperature field for concrete slab

Create instances:

-slab



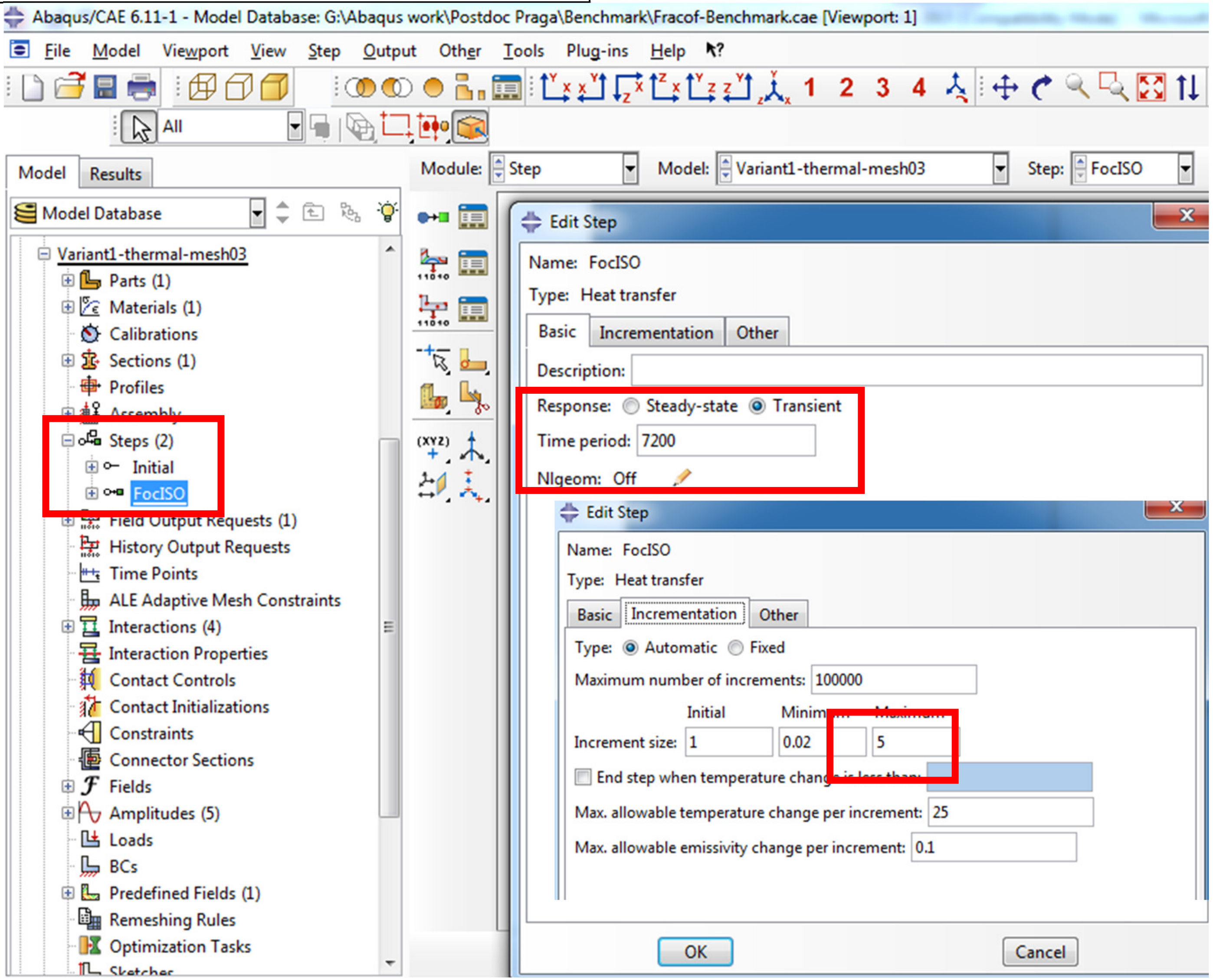
Numerical model

Temperature field for concrete slab

Define heat transfer step:

-transient

-time



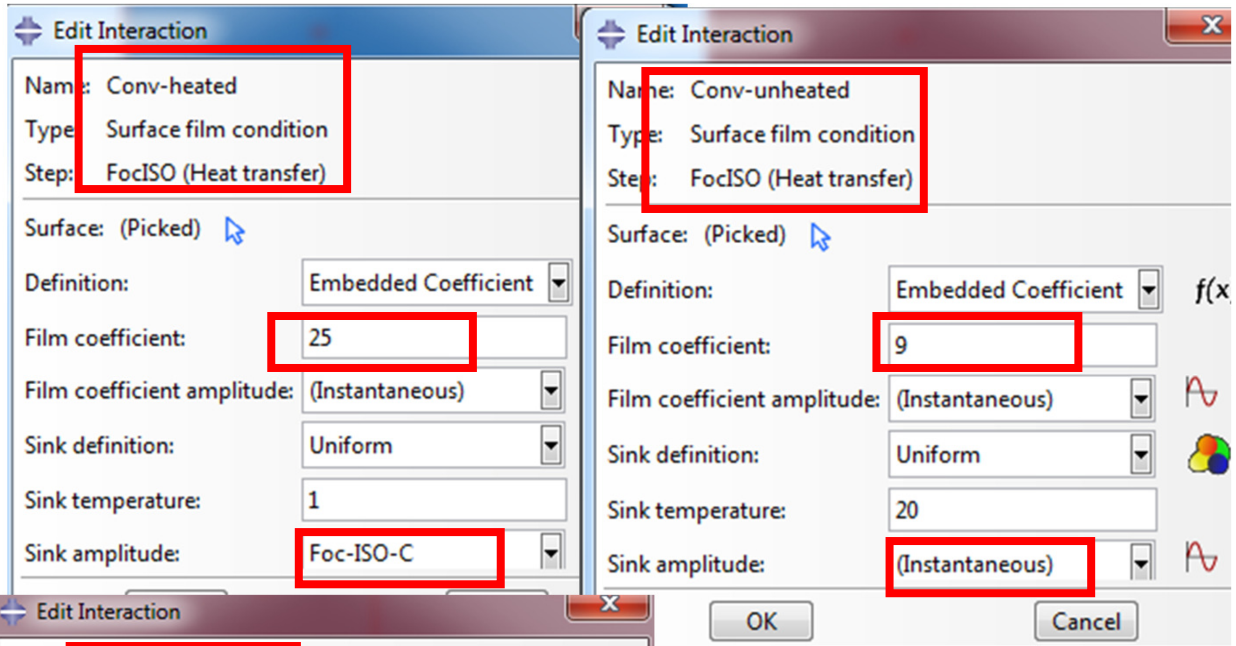
Numerical model

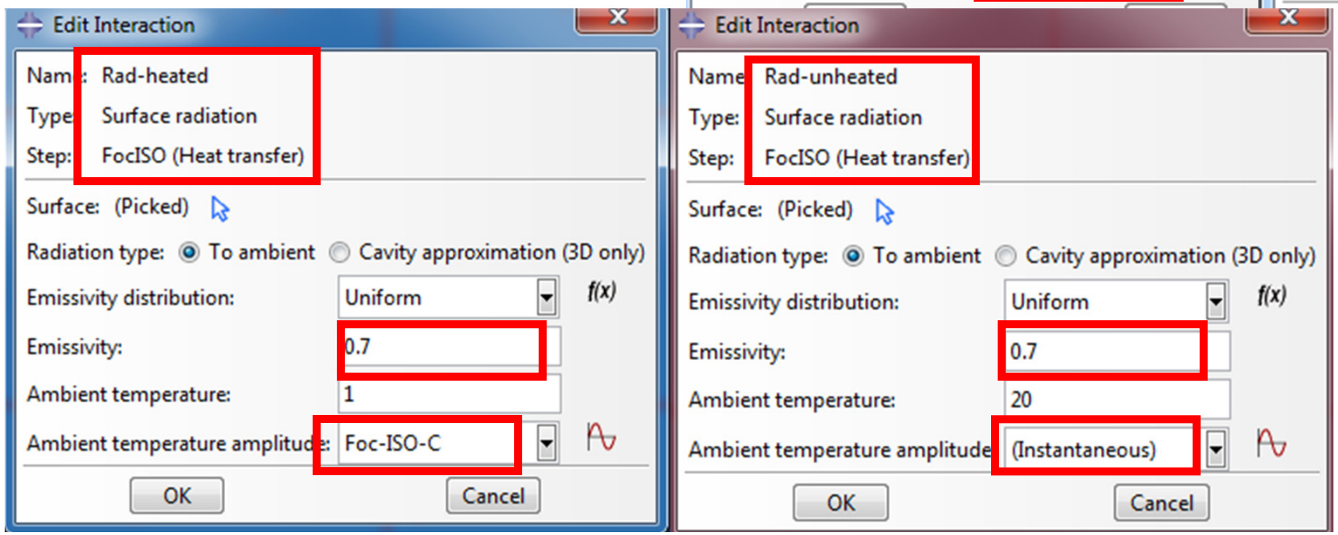
Temperature field for concrete slab

Define interactions:

-convection for heated and unheated sides

-radiation for heated and unheated side



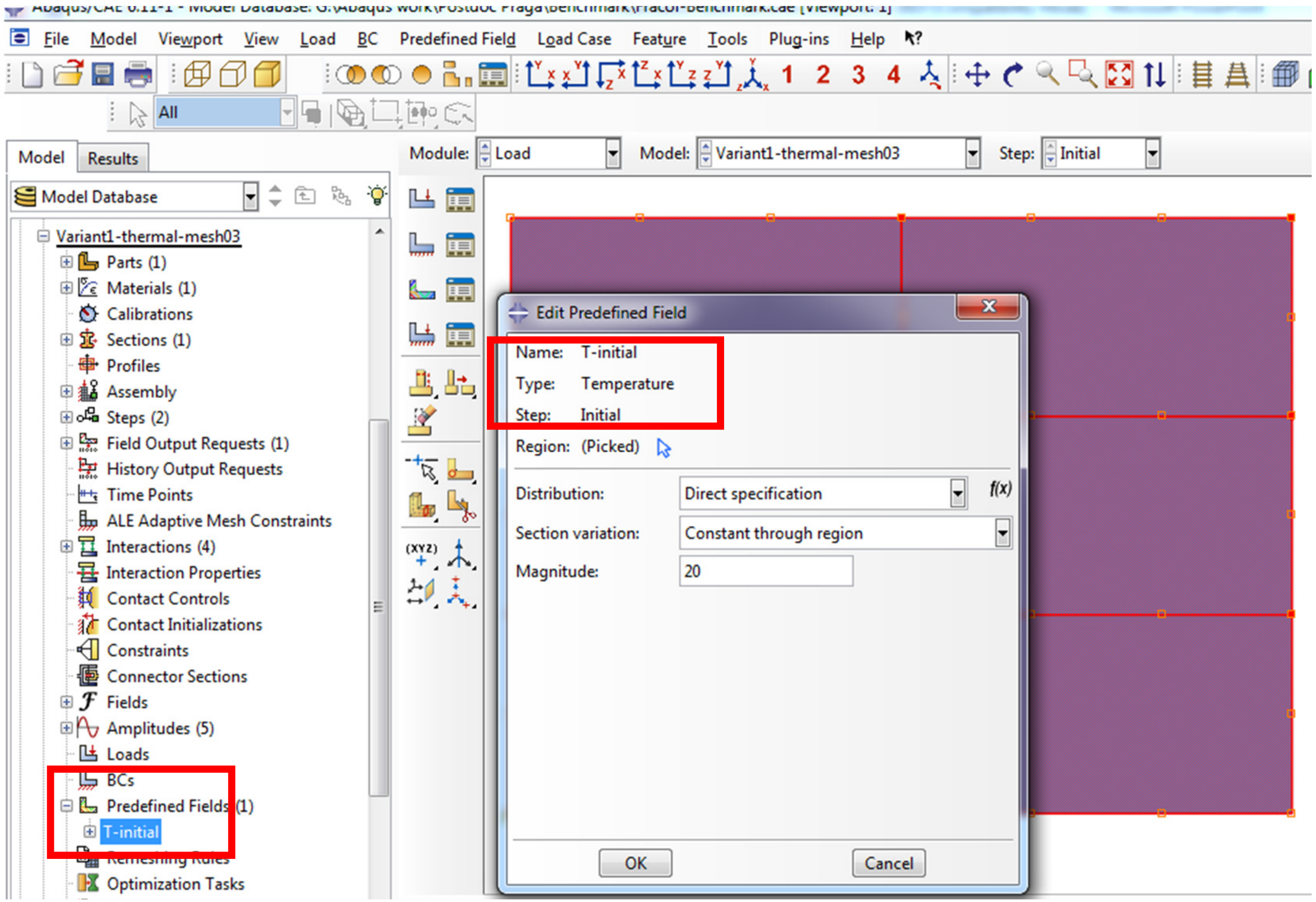


Numerical model

Temperature field for concrete slab

Define initial temperature:

-predefined field – constant through region

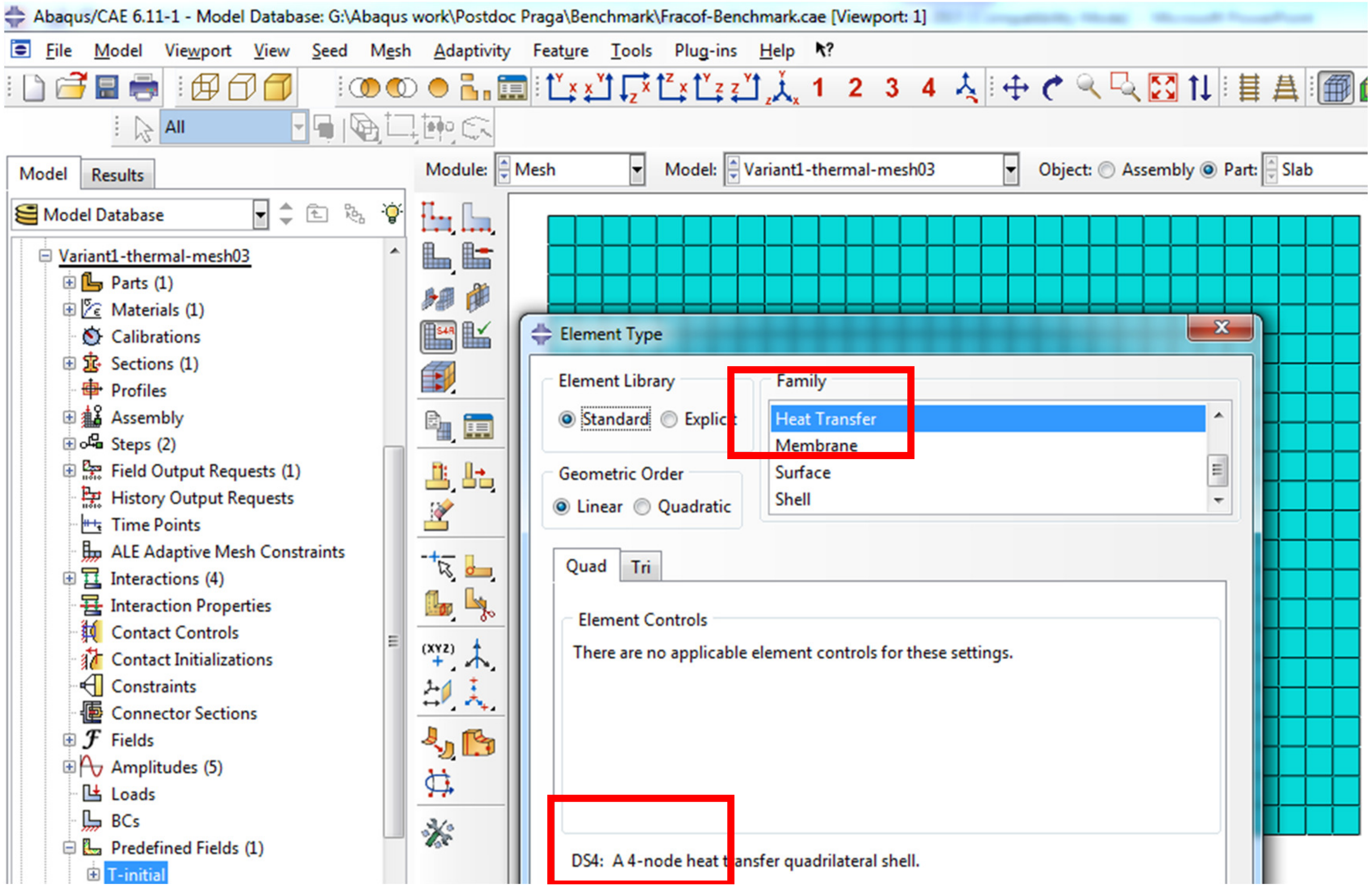


Numerical model

Temperature field for concrete slab

Define mesh and finite elements:

DS4 (0.3 m)

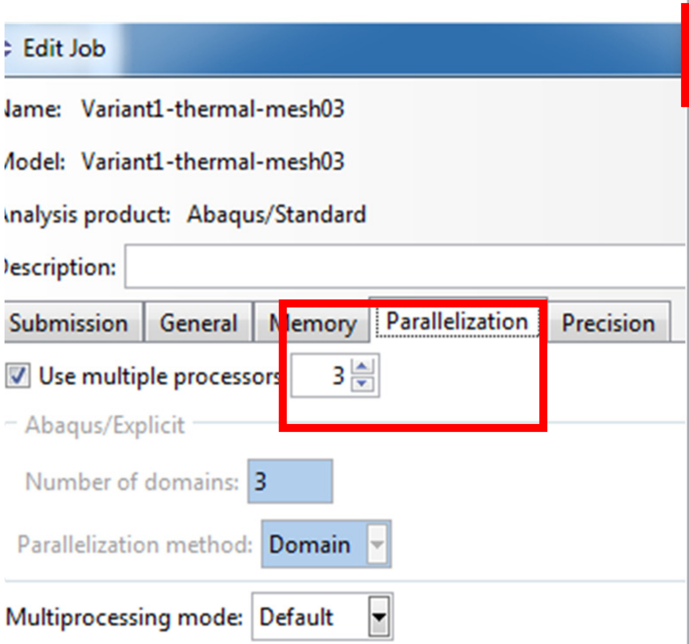


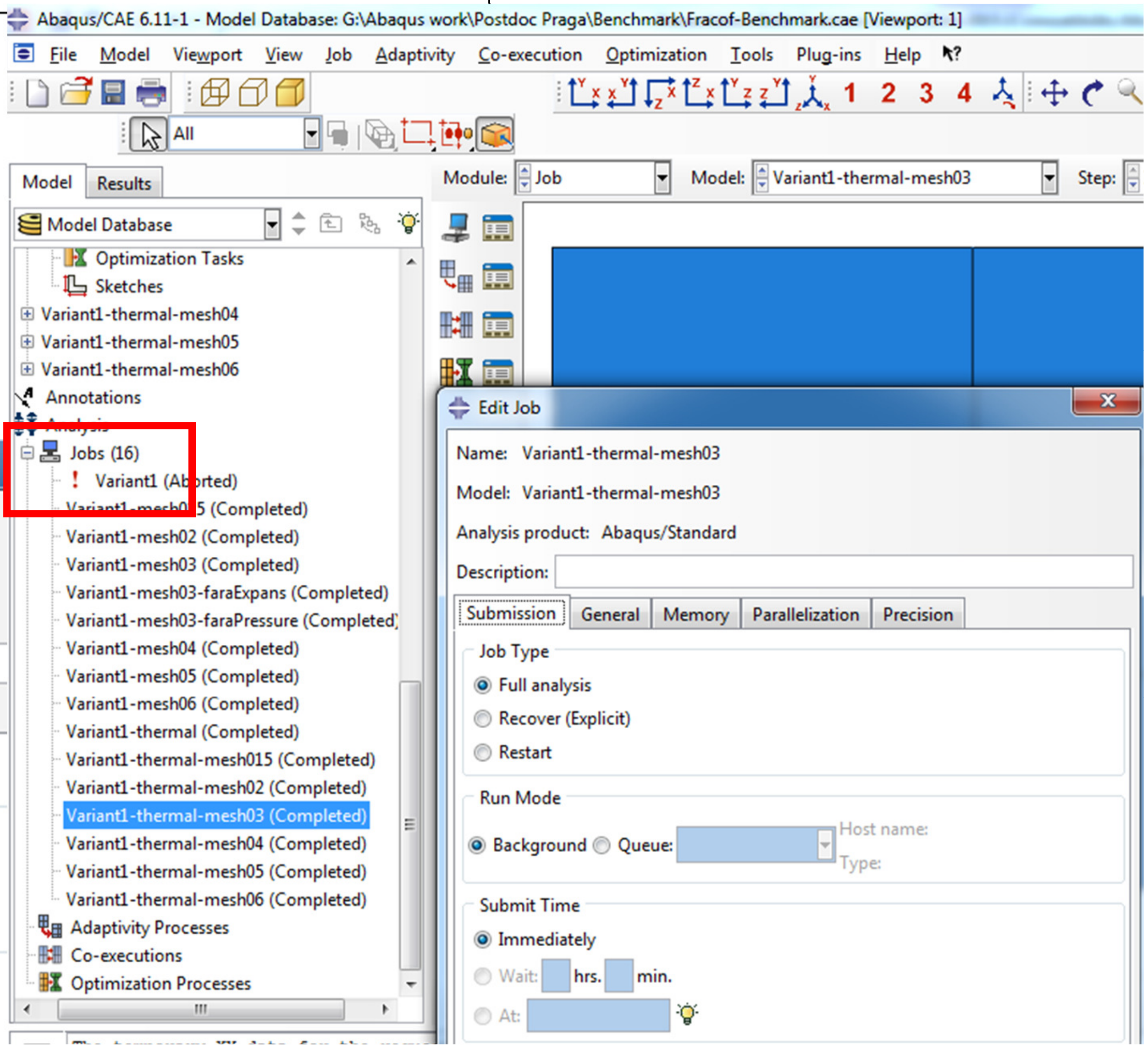
Numerical model

Temperature field for concrete slab

Run analysis:

Create job and submit



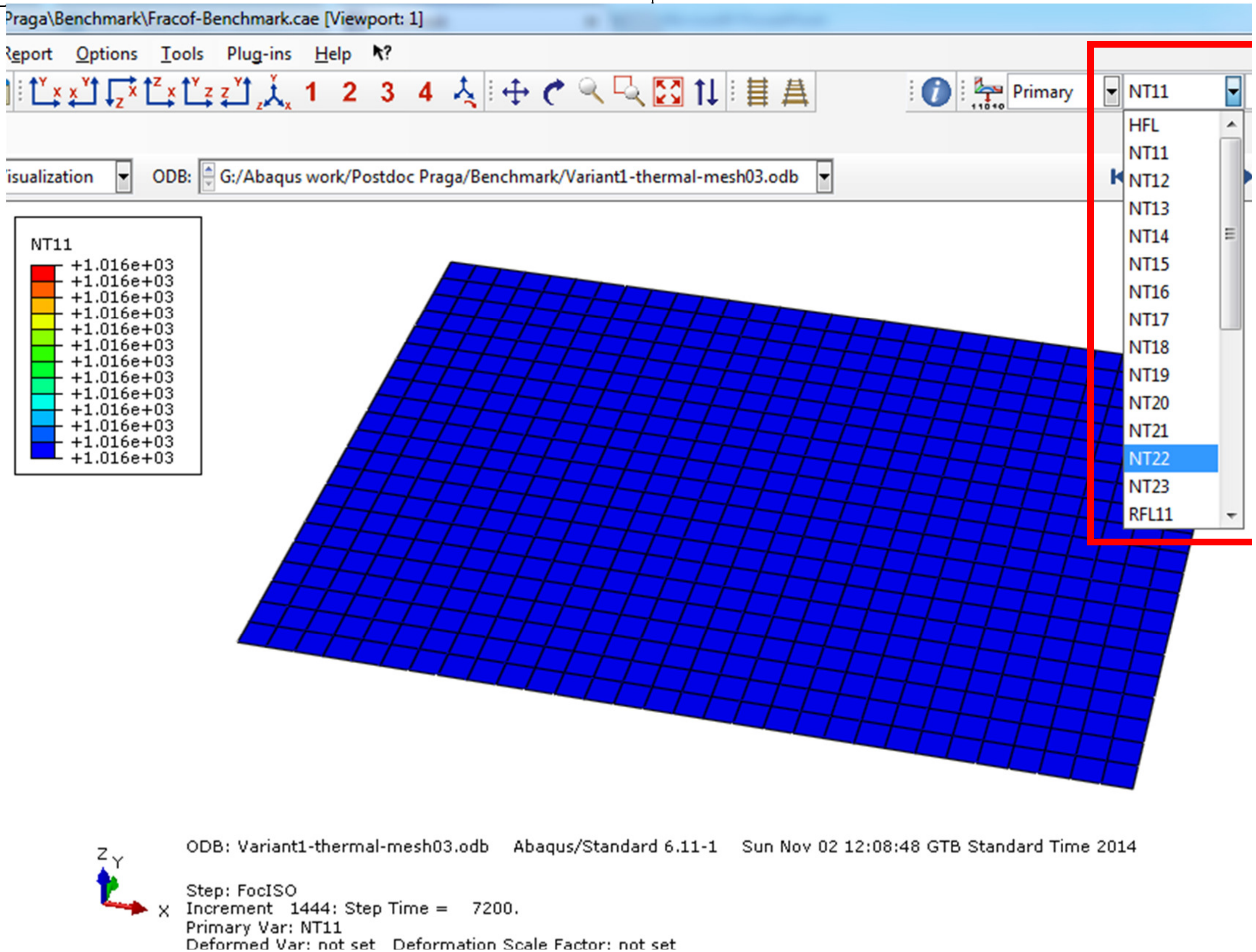


Numerical model

Temperature field for concrete slab

Results:

Nodal temperatures

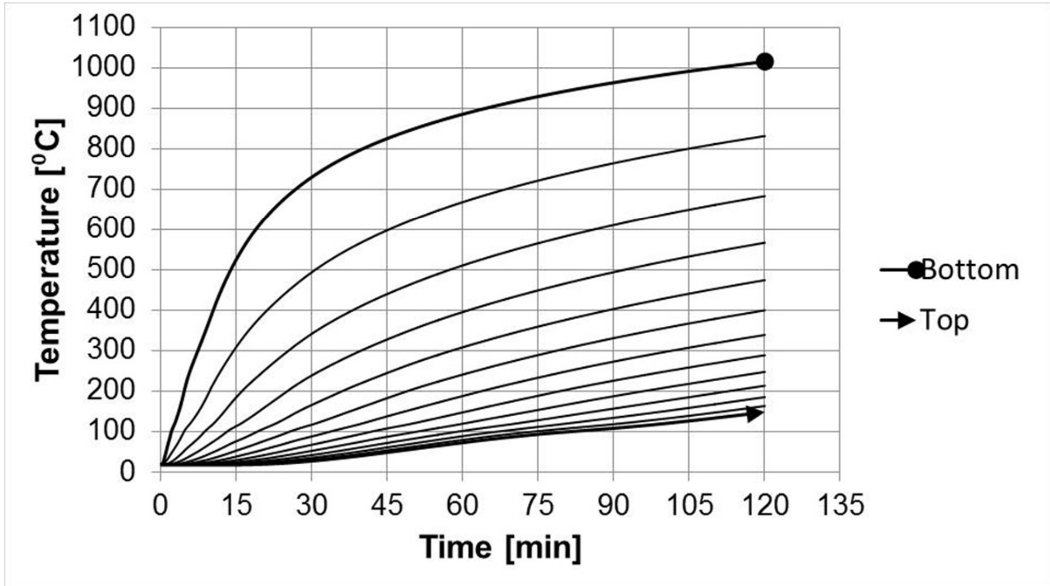


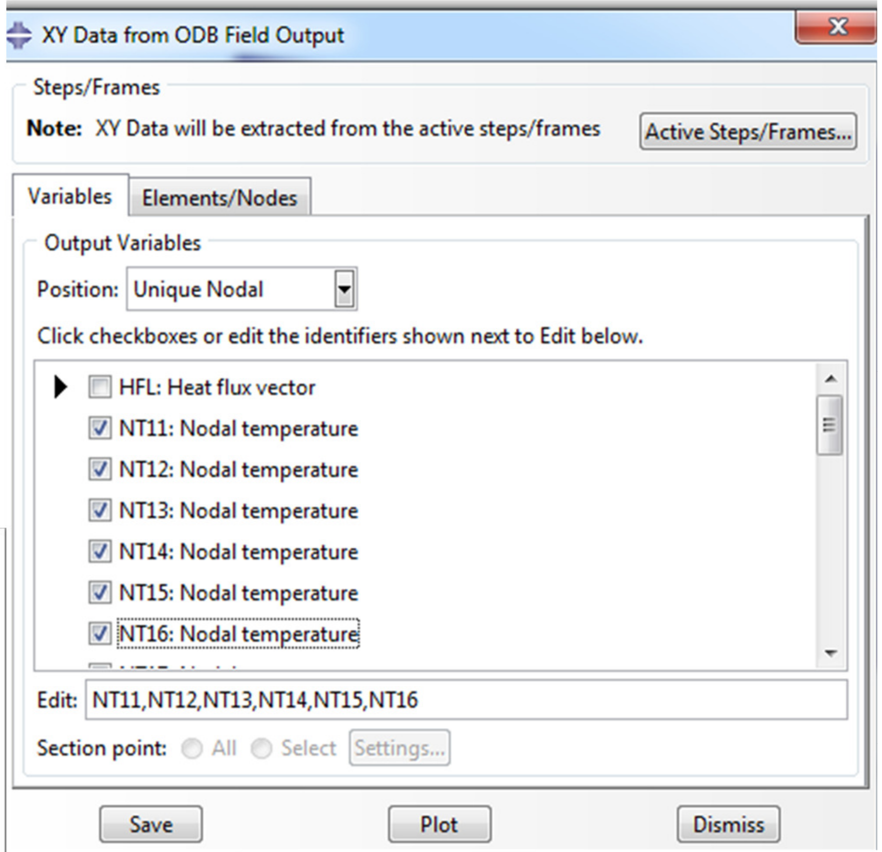
Numerical model

Temperature field for concrete slab

Results:

Nodal temperatures





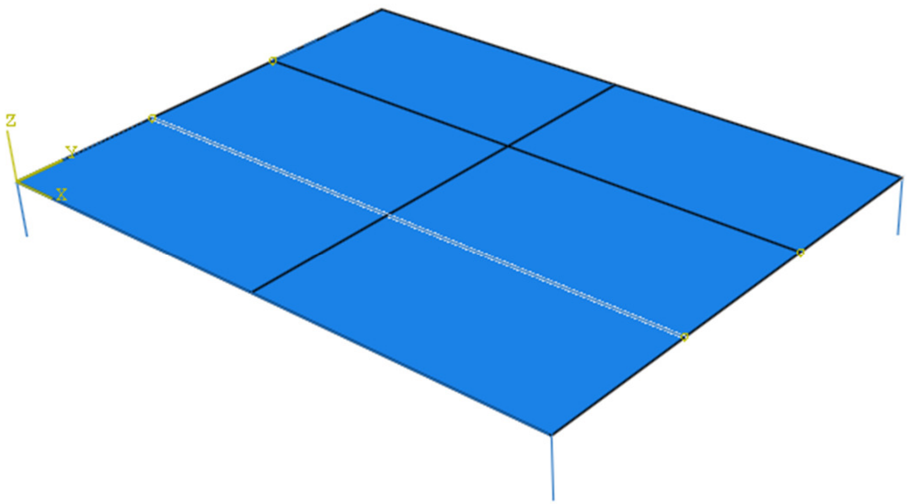
Numerical model

Structural analysis of composite slab

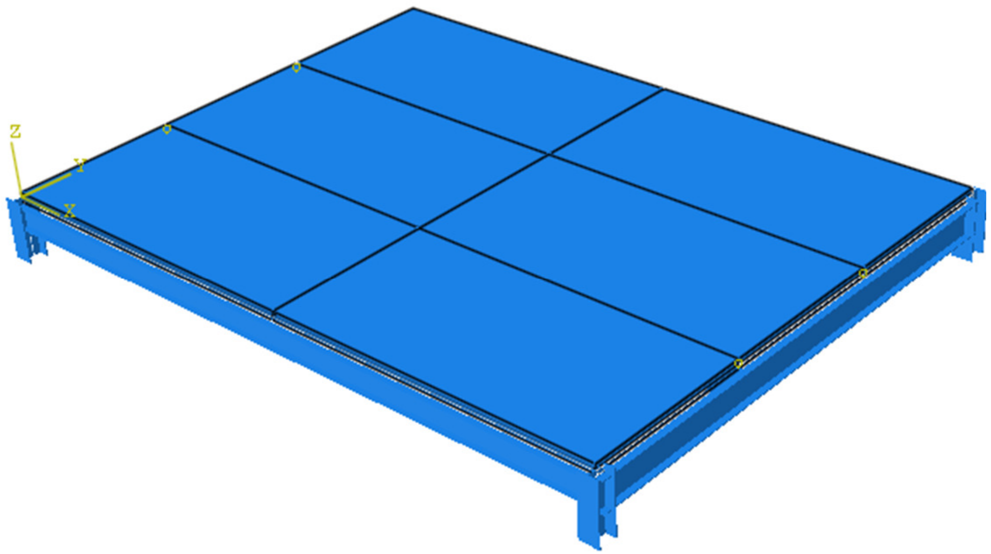
Starts from a saved as model of thermal analysis of concrete slab

All structural elements, beams and columns, are defined as linear *wire element*

Wireframe



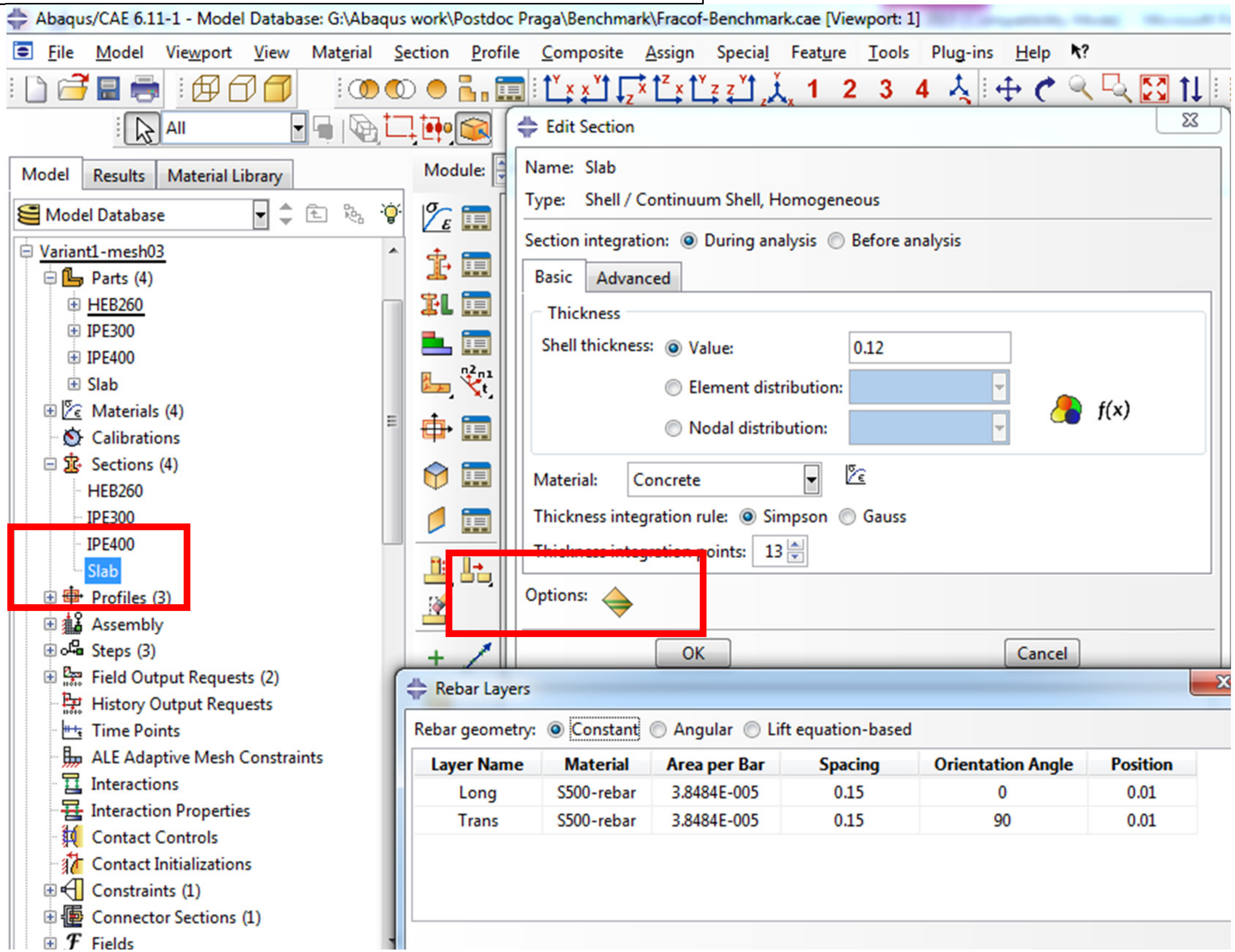
Rendered view



Numerical model

Structural analysis of composite slab

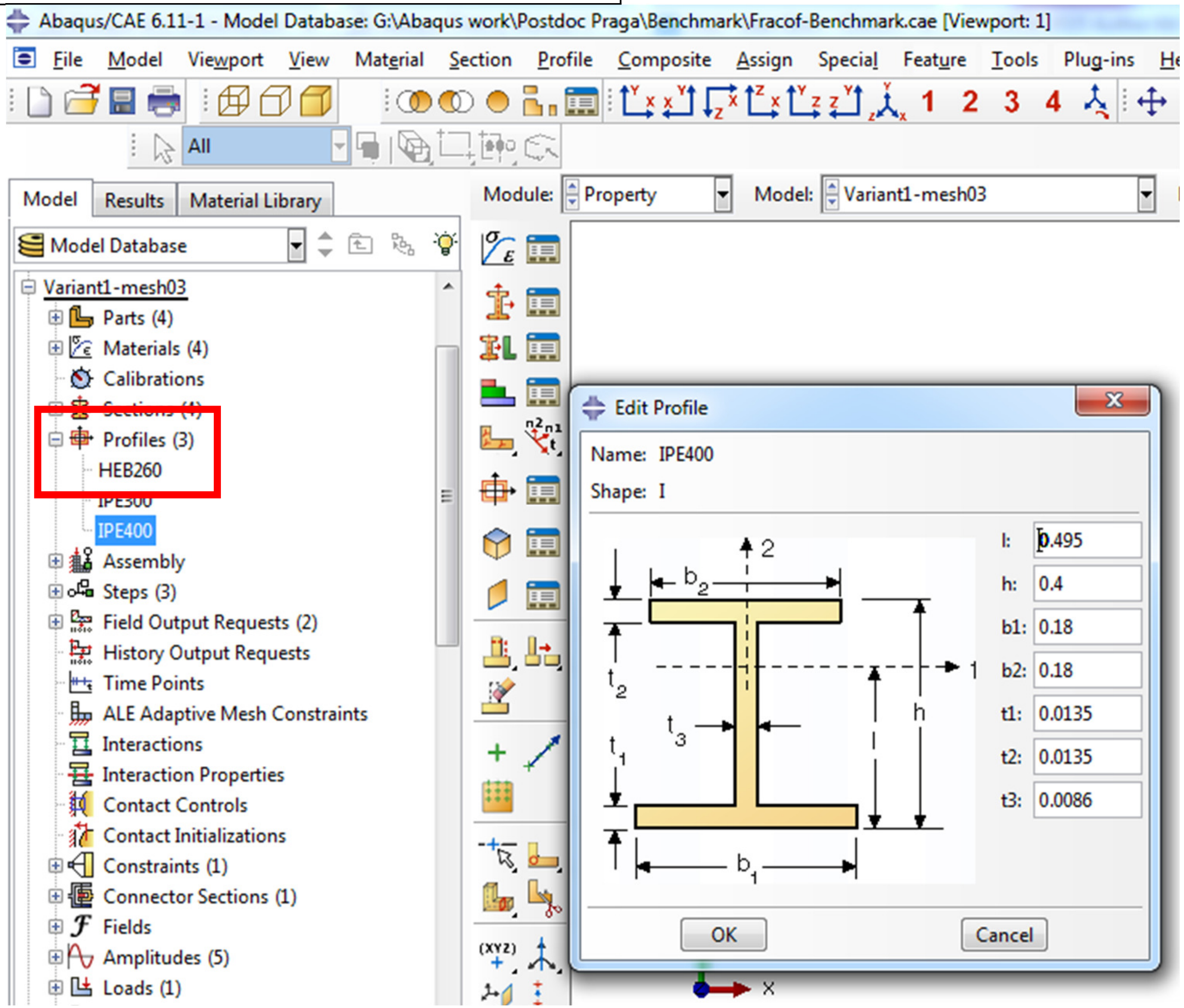
Add the reinforcement of the concrete slab



Numerical model

Structural analysis of composite slab

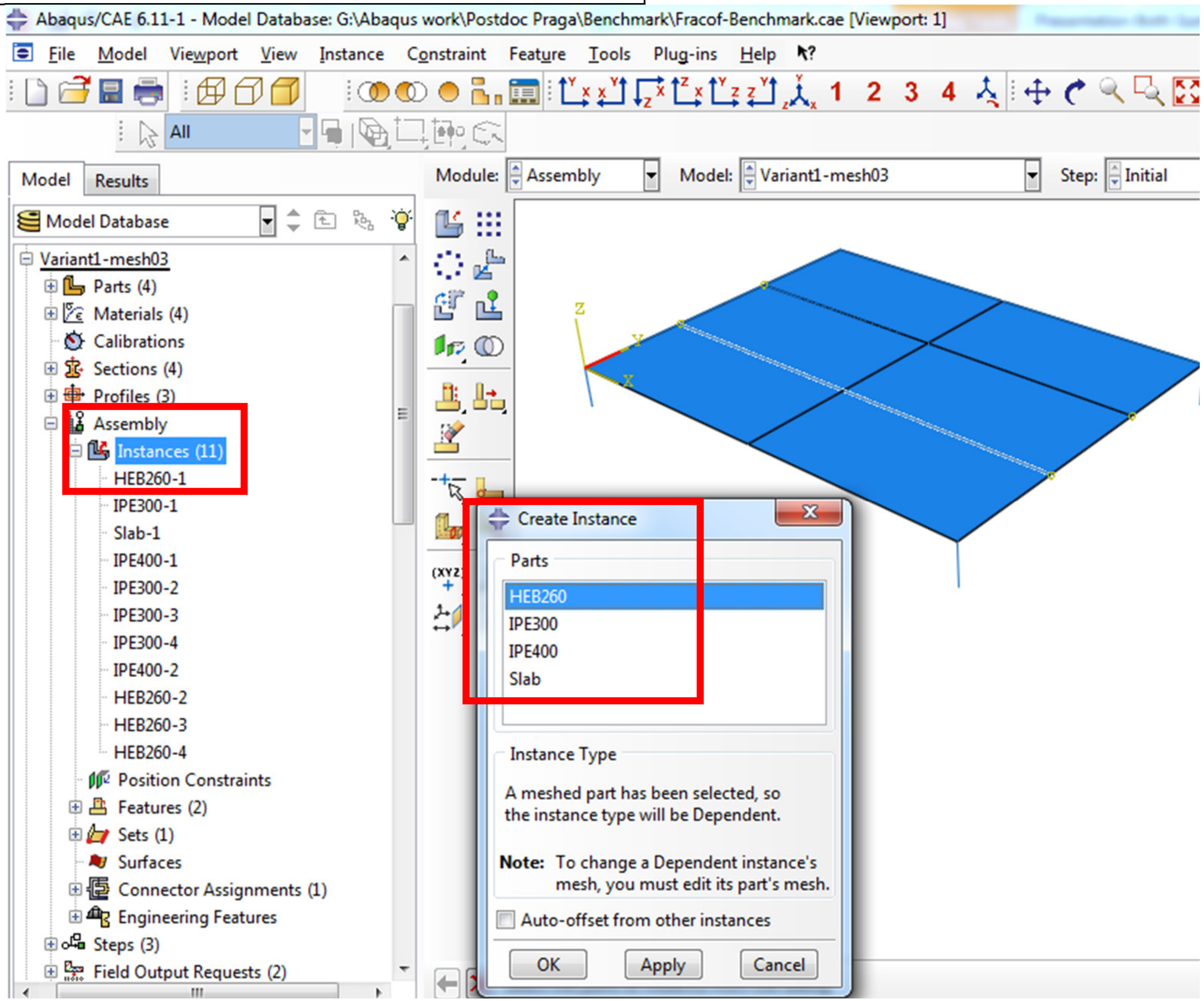
Define “profiles” for the wire elements and orientation



Numerical model

Structural analysis of composite slab

Create "instances" from parts for all elements and "construct" the structure



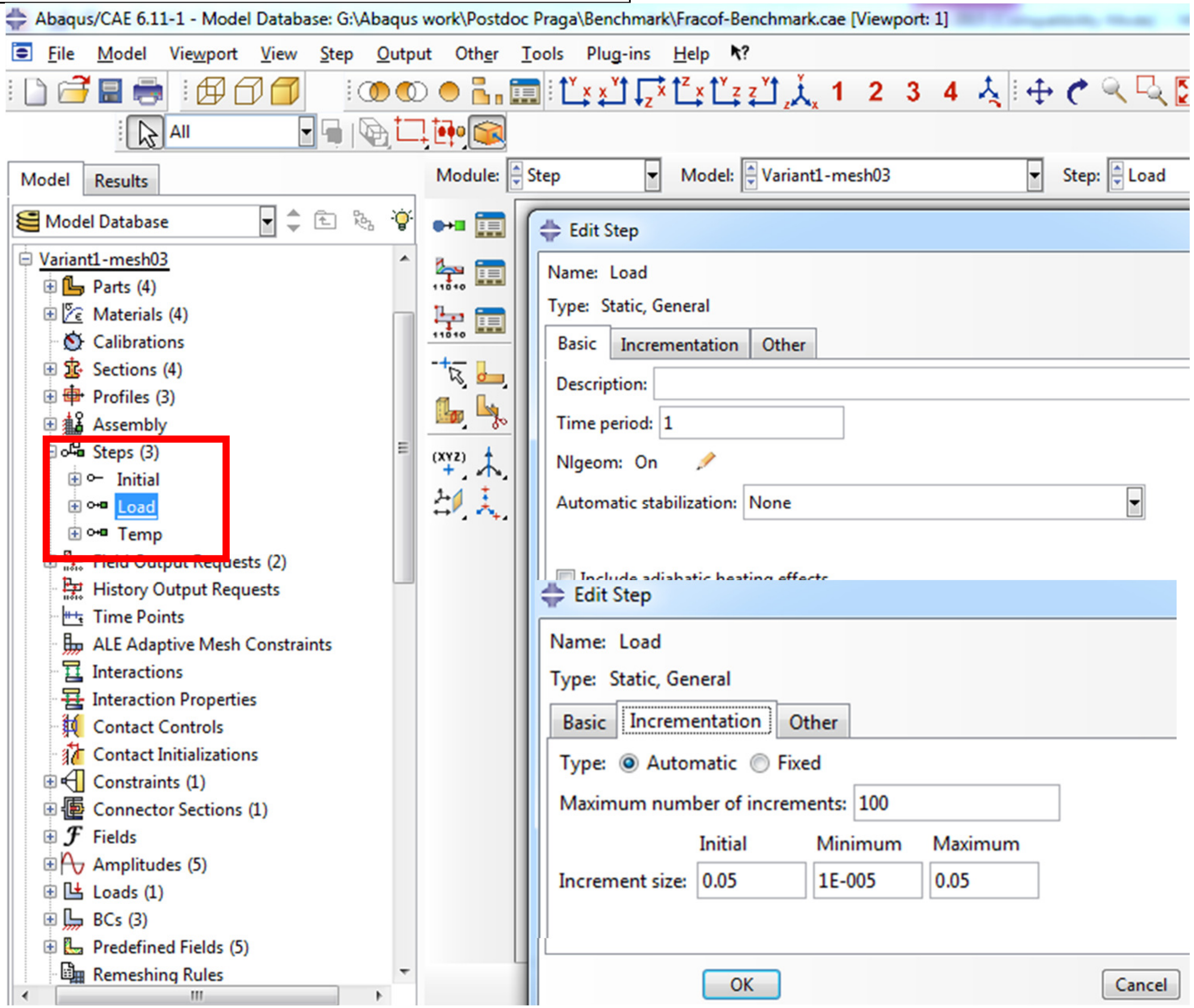
Numerical model

Structural analysis of composite slab

Define steps for analysis:

-for mechanical loading

-for temperature influence



Both steps are "Static, General"

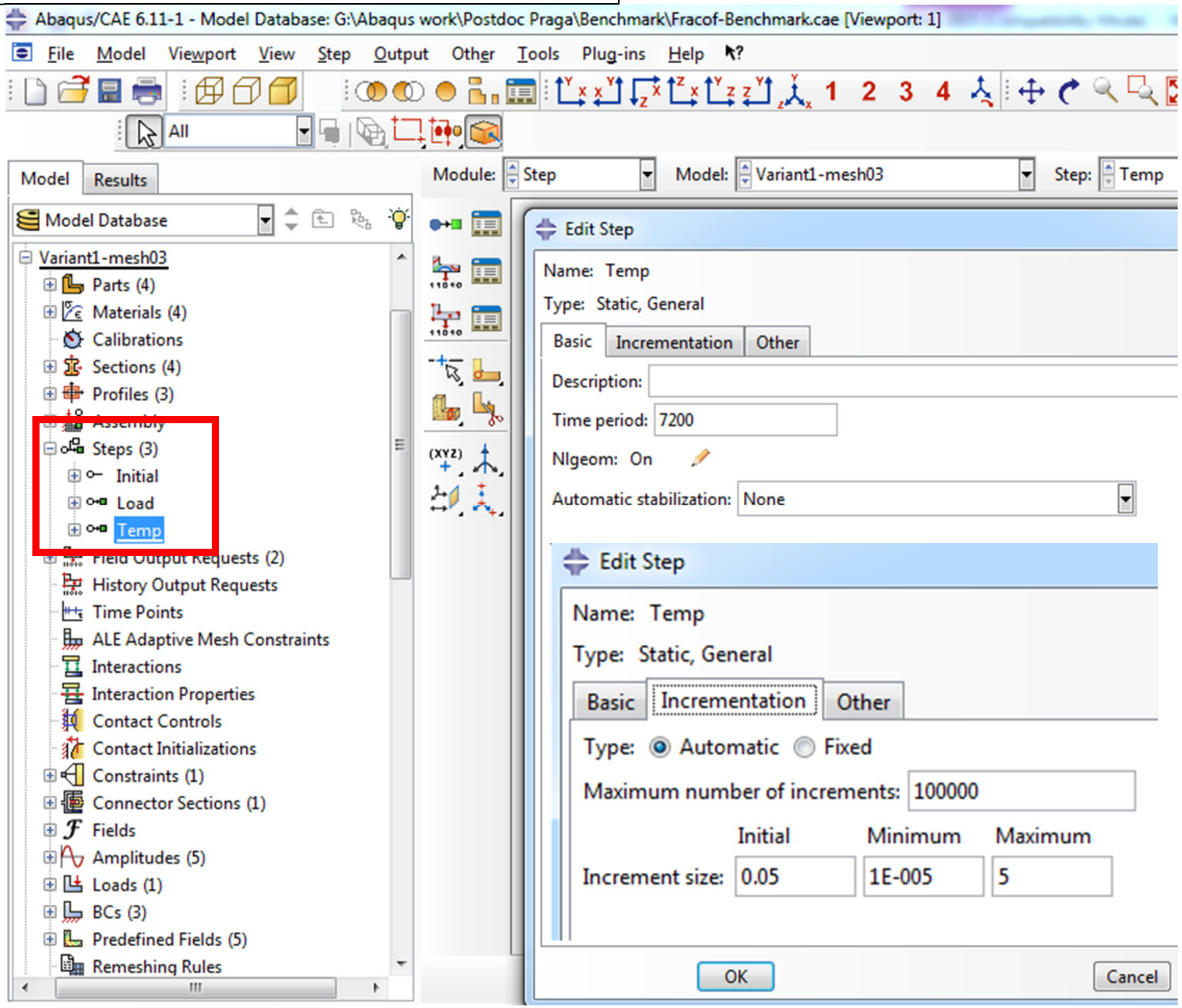
Numerical model

Structural analysis of composite slab

Define steps for analysis:

-for mechanical loading

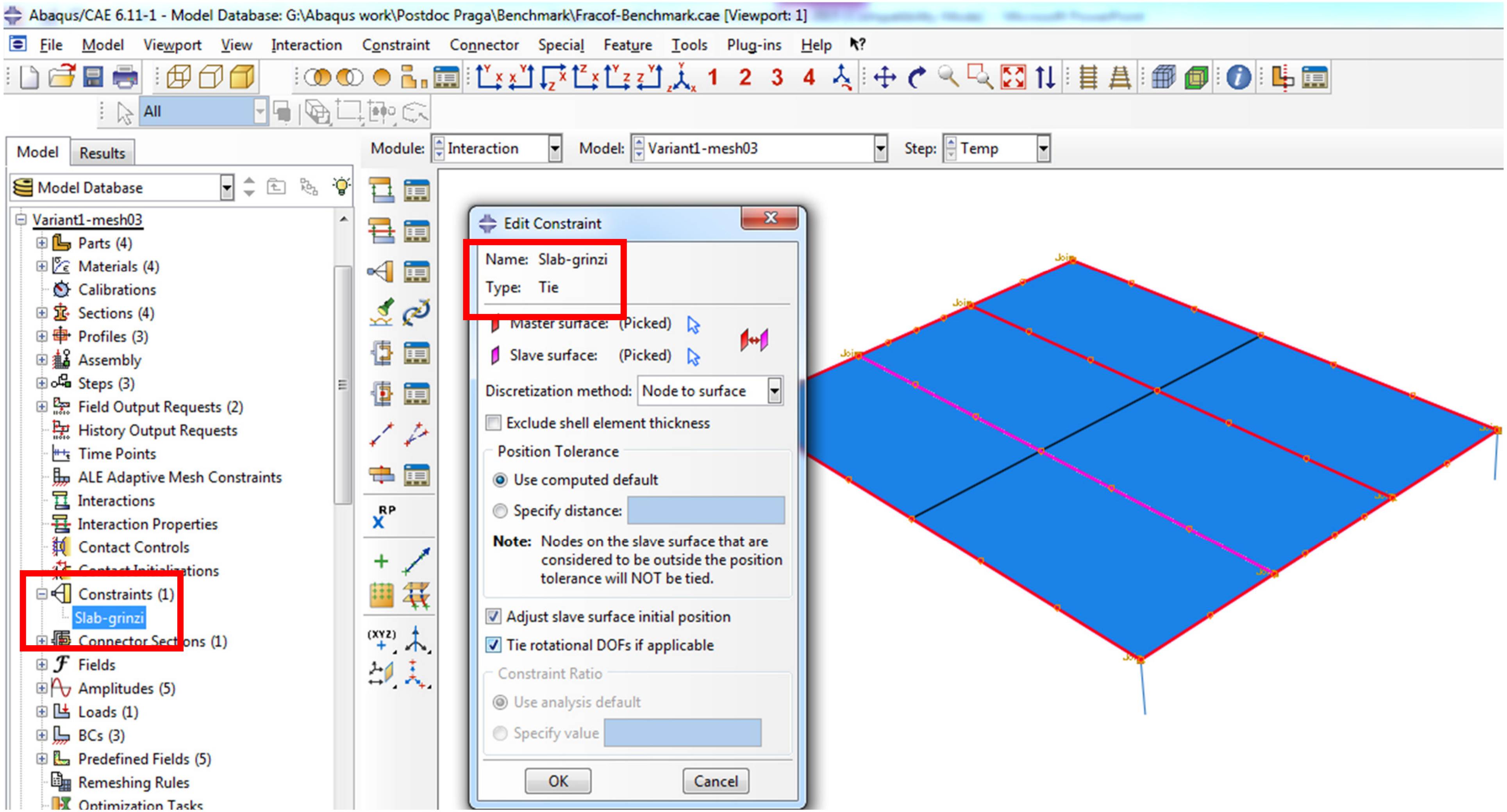
-for temperature influence



Numerical model

Structural analysis of composite slab

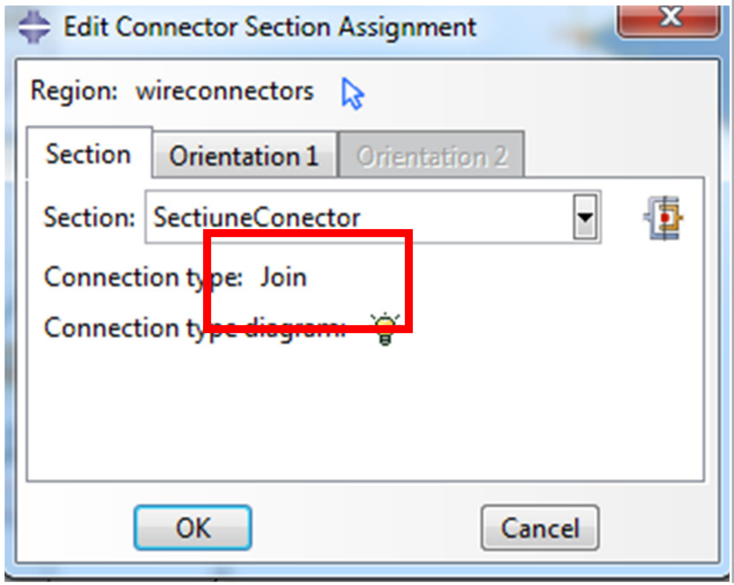
Define mechanical interactions between slab and beams: *constraints*

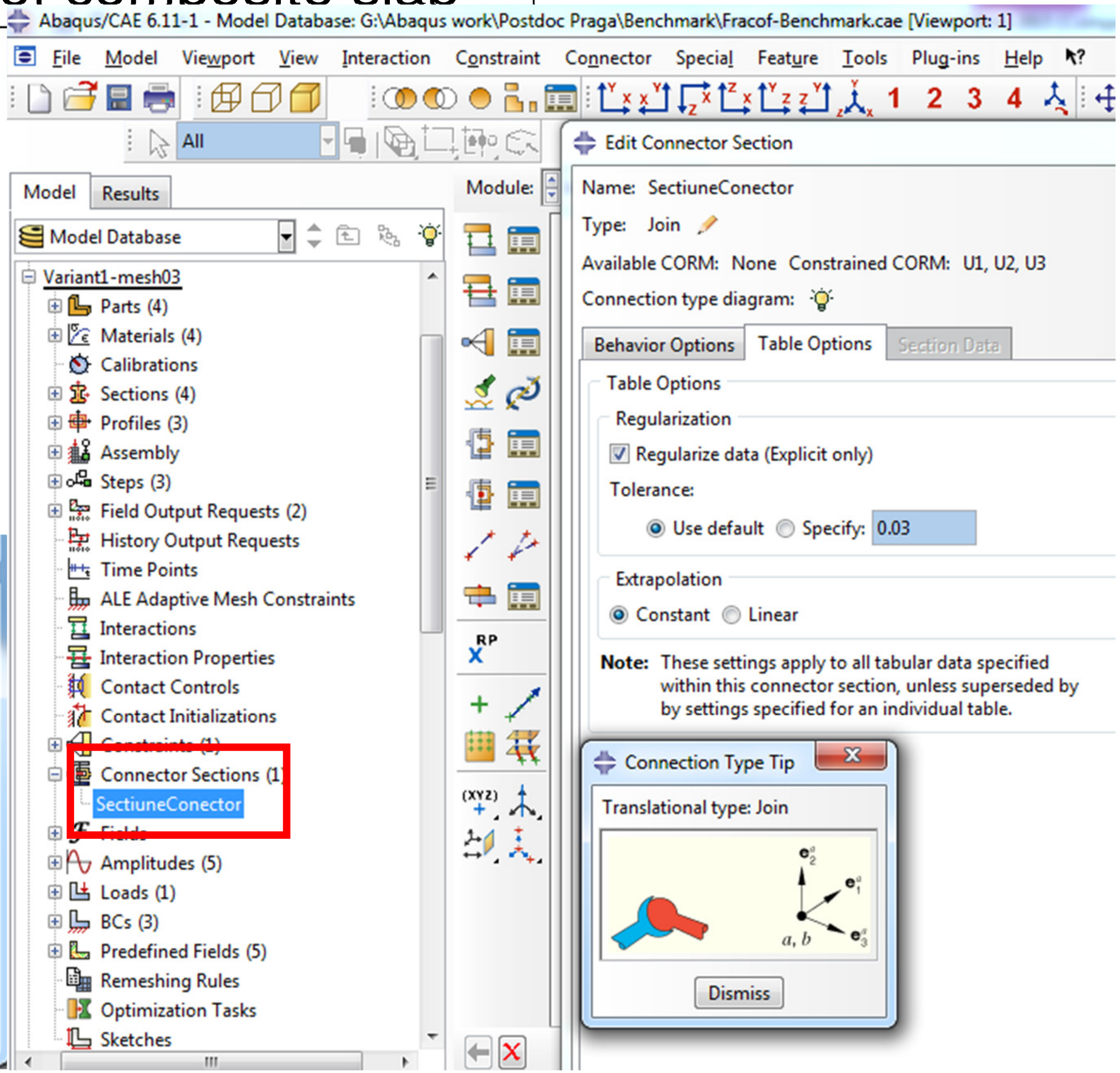


Numerical model

Structural analysis of composite slab

Define mechanical interactions of connections: *connector section "join"*





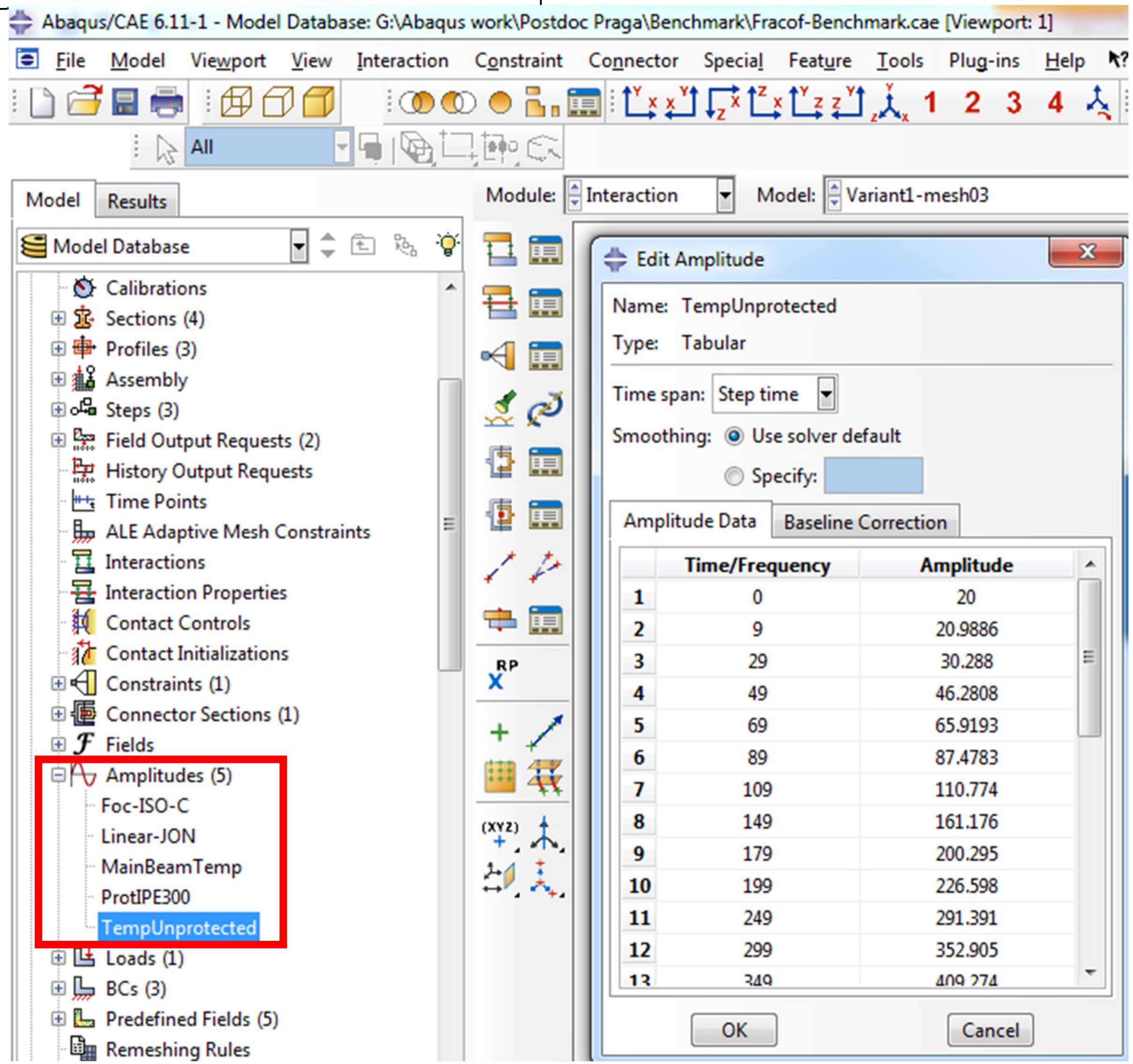
Numerical model

Structural analysis of composite slab

Define *Amplitudes* :

-temperature of unprotected beams,

-variation of reference lines for protected primary and secondary beams.

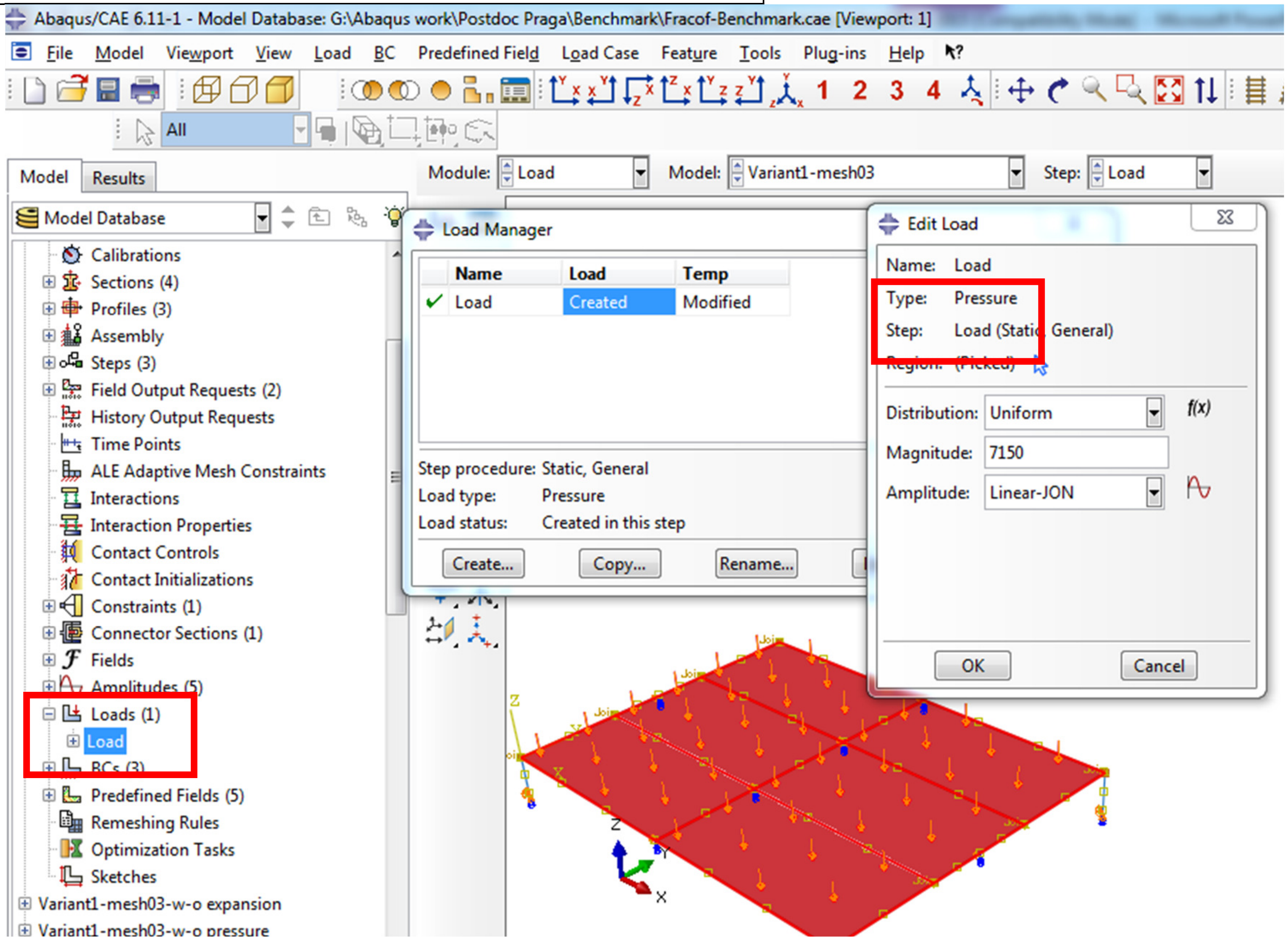


Numerical model

Structural analysis of composite slab

Define *Loads*

-Pressure.



Numerical model

Structural analysis of composite slab

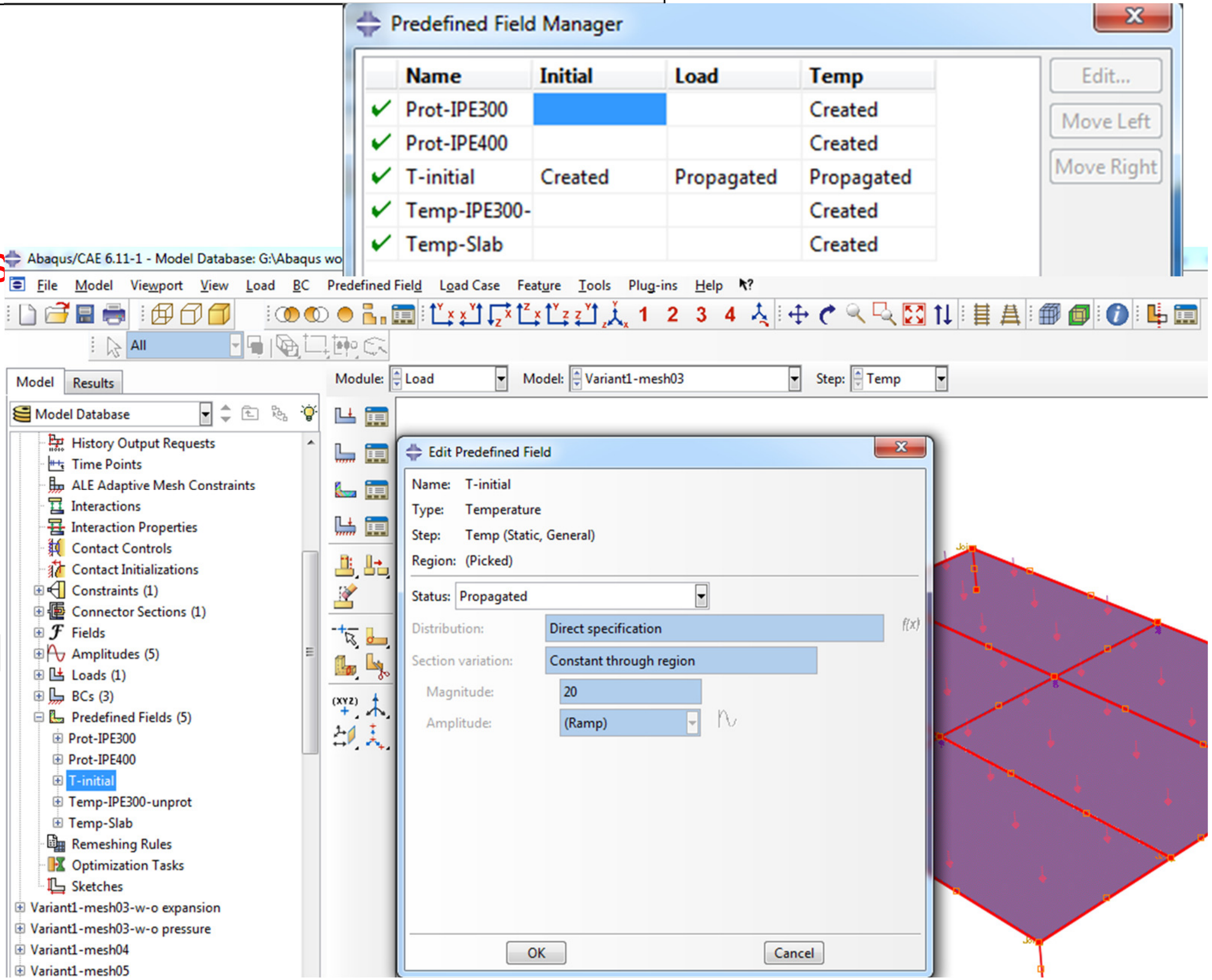
Predefined fields:

-Initial temperatures (entire structure)

-Gradients through beam section (protected beams)

-Constant through region (unprotected beam)

-From thermal analysis data base (slab)



Numerical model

Structural analysis of composite slab

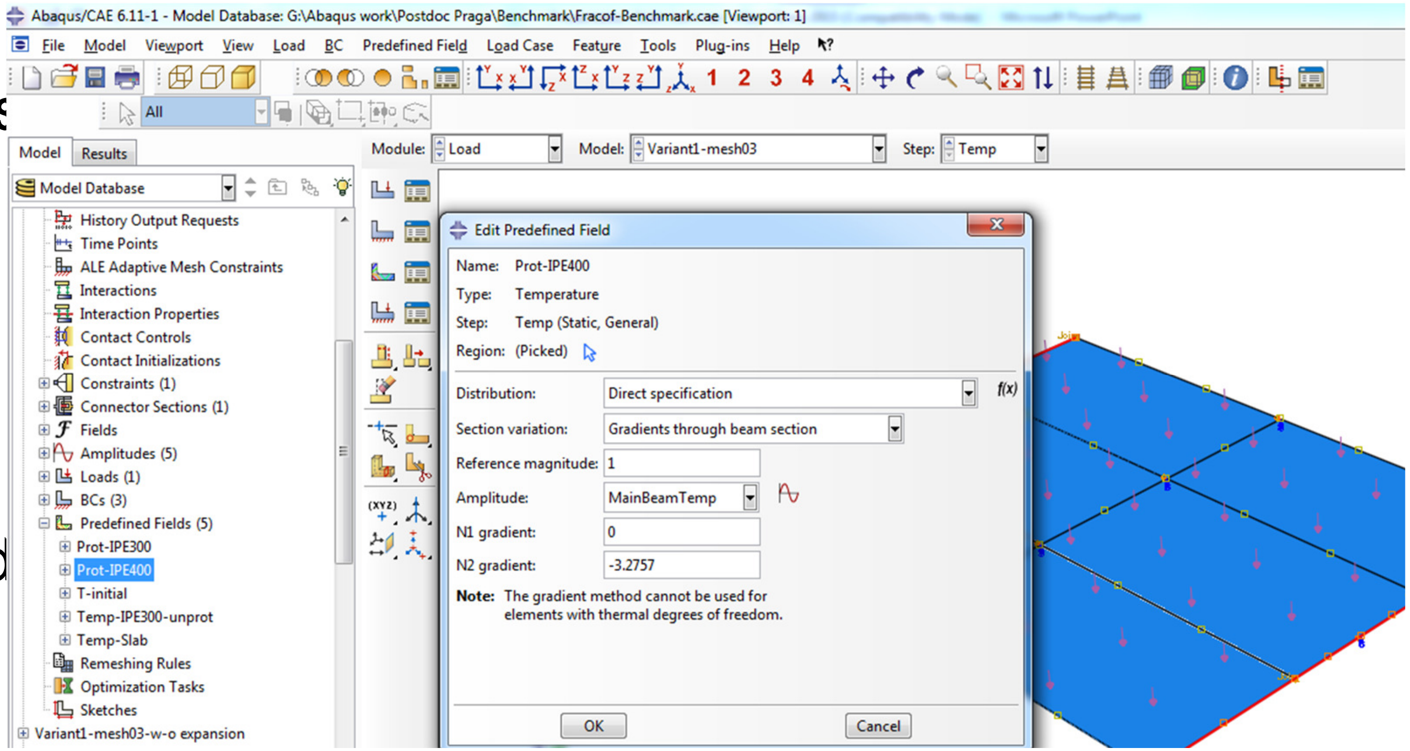
Predefined fields:

-Initial temperatures (entire structure)

-Gradients through beam section (protected beams)

-Constant through region (unprotected beam)

-From thermal analysis data base (slab)



Numerical model

Structural analysis of composite slab

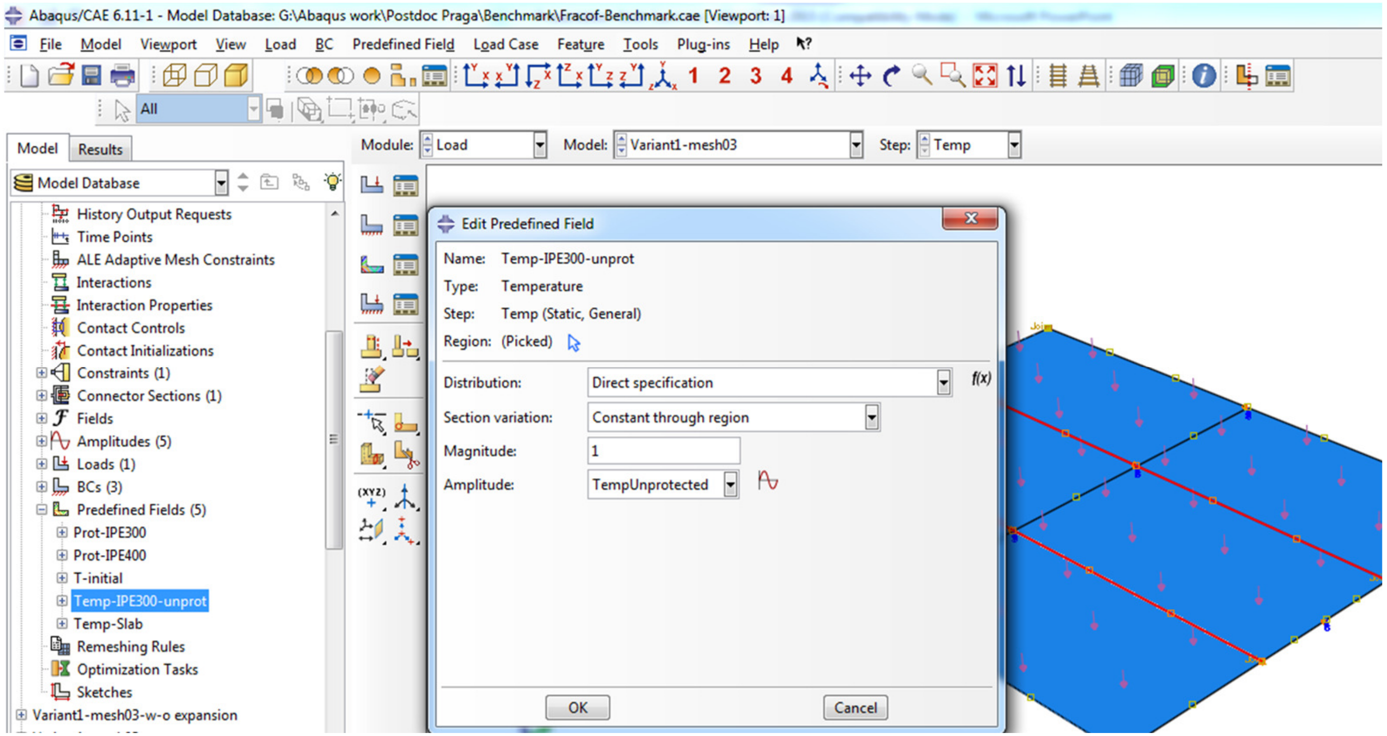
Predefined fields:

-Initial temperatures (entire structure)

-Gradients through beam section (protected beams)

-Constant through region (unprotected beam)

-From thermal analysis data base (slab)



Numerical model

Structural analysis of composite slab

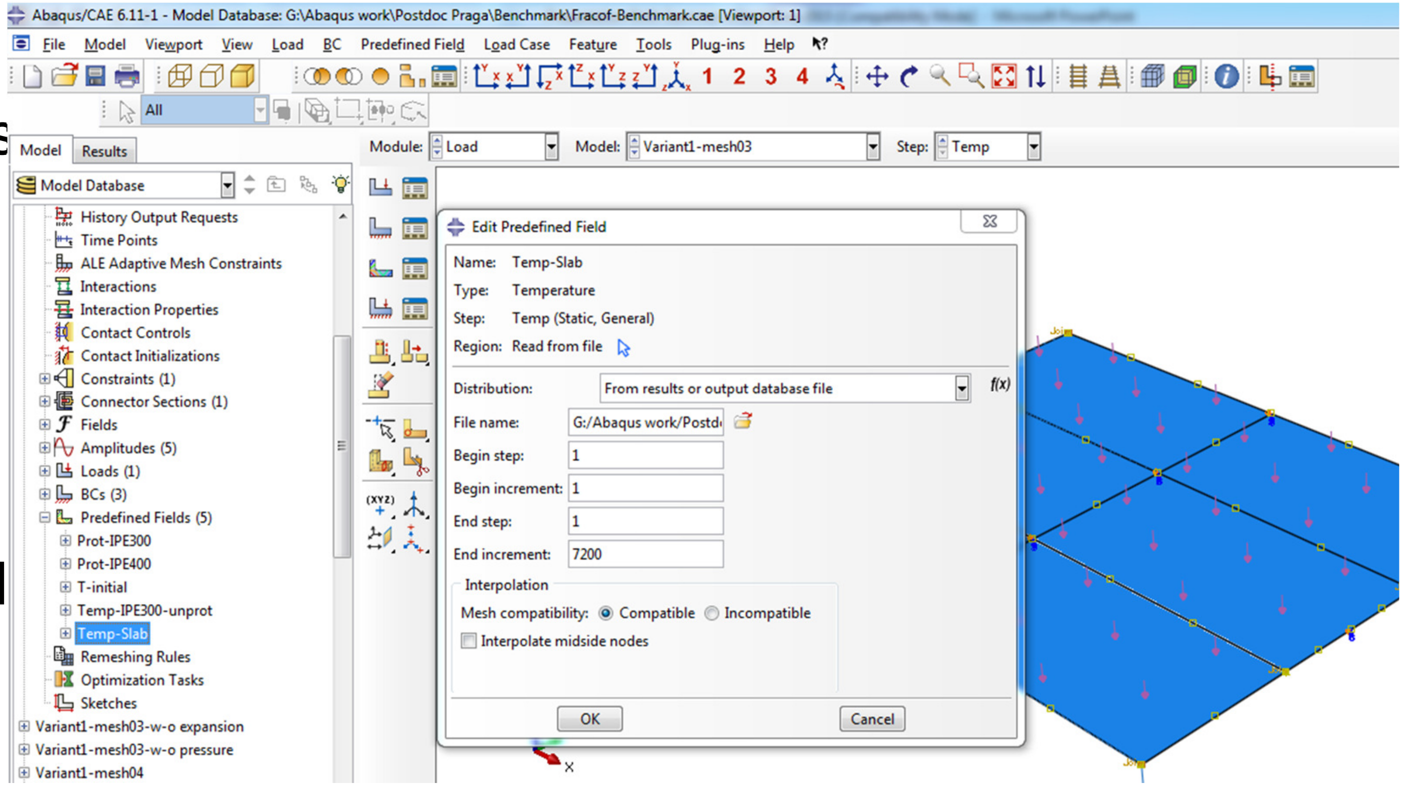
Predefined fields:

-Initial temperatures (entire structure)

-Gradients through beam section (protected beams)

-Constant through region (unprotected beam)

-From thermal analysis data base (slab)



Numerical model

Structural analysis of composite slab

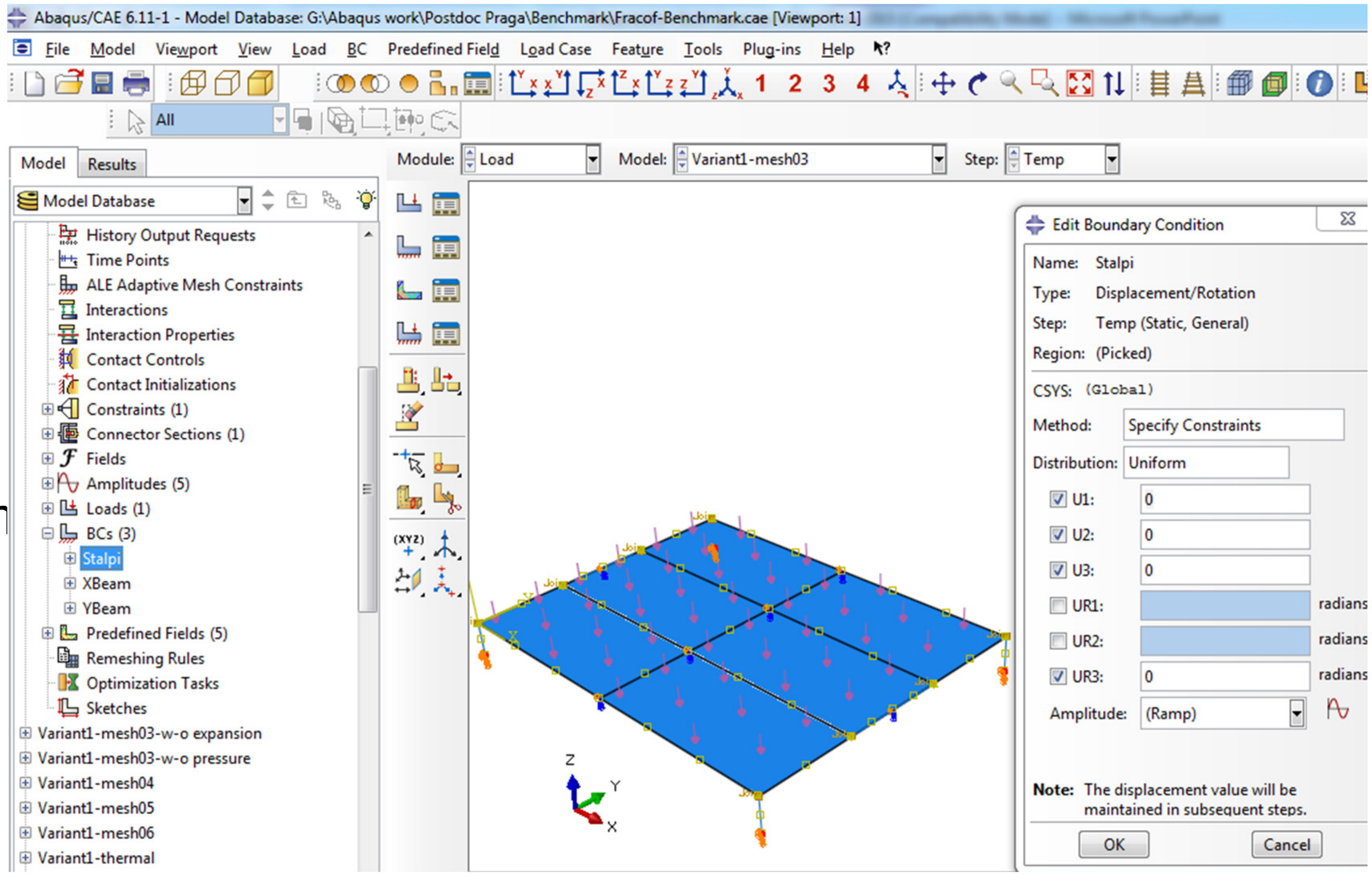
Support conditions:

Boundary conditions:

- columns

-longit. direction

-transv. direction



Numerical model

Structural analysis of composite slab

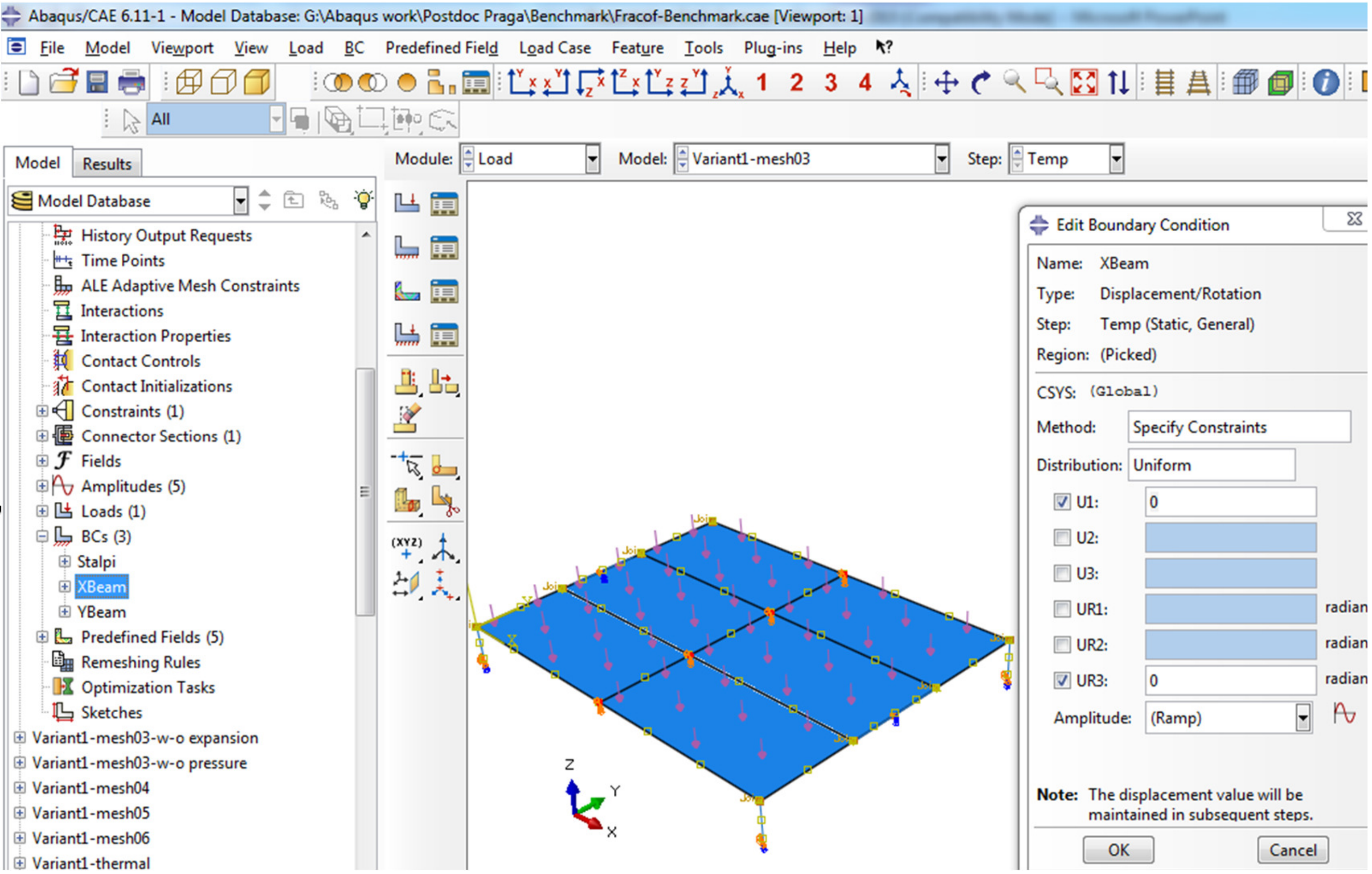
Support conditions:

Boundary conditions:

- columns

-longit. direction

-transv. direction



Numerical model

Structural analysis of composite slab

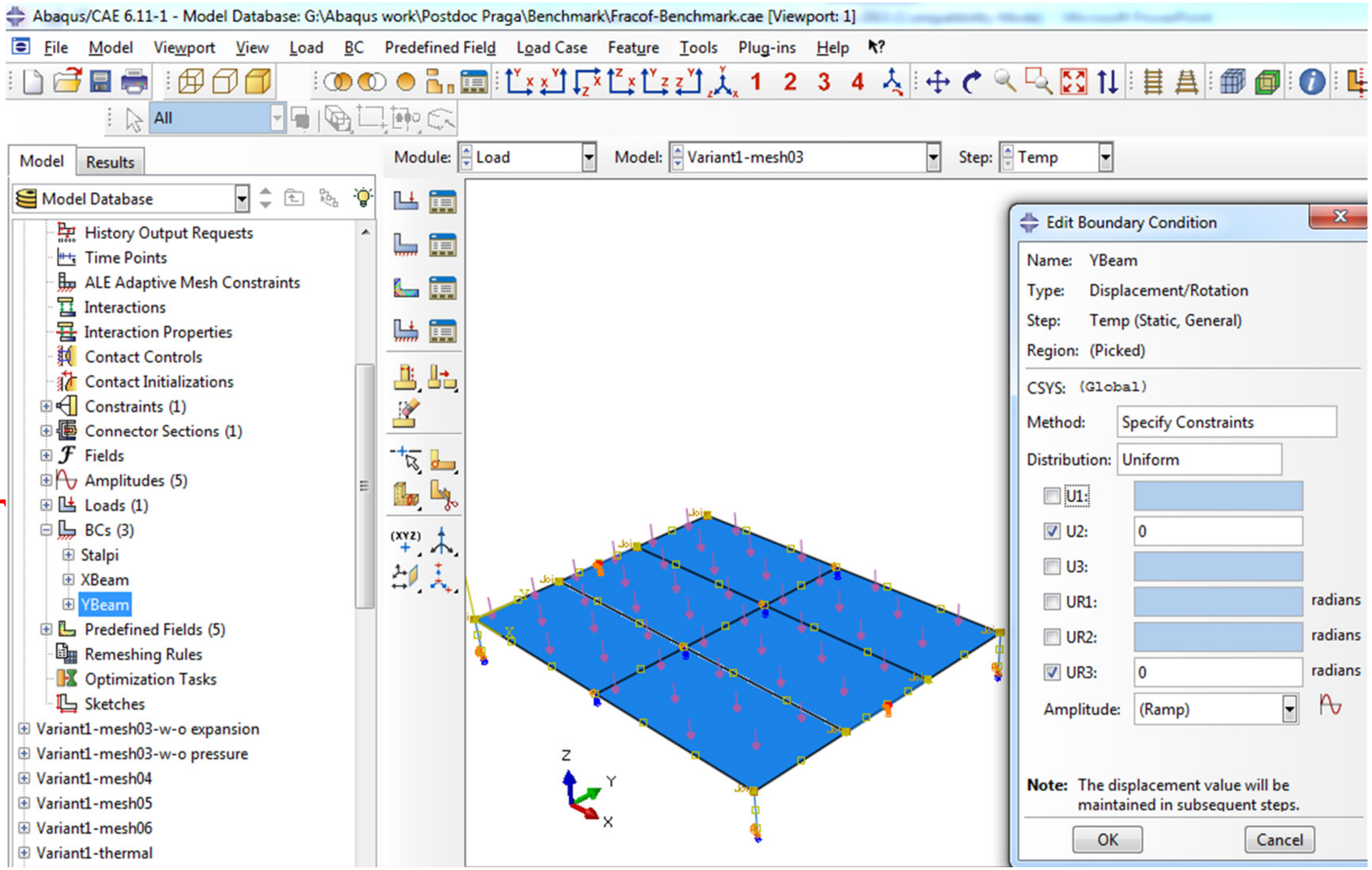
Support conditions:

Boundary conditions:

- columns

-longit. direction

-transv. direction

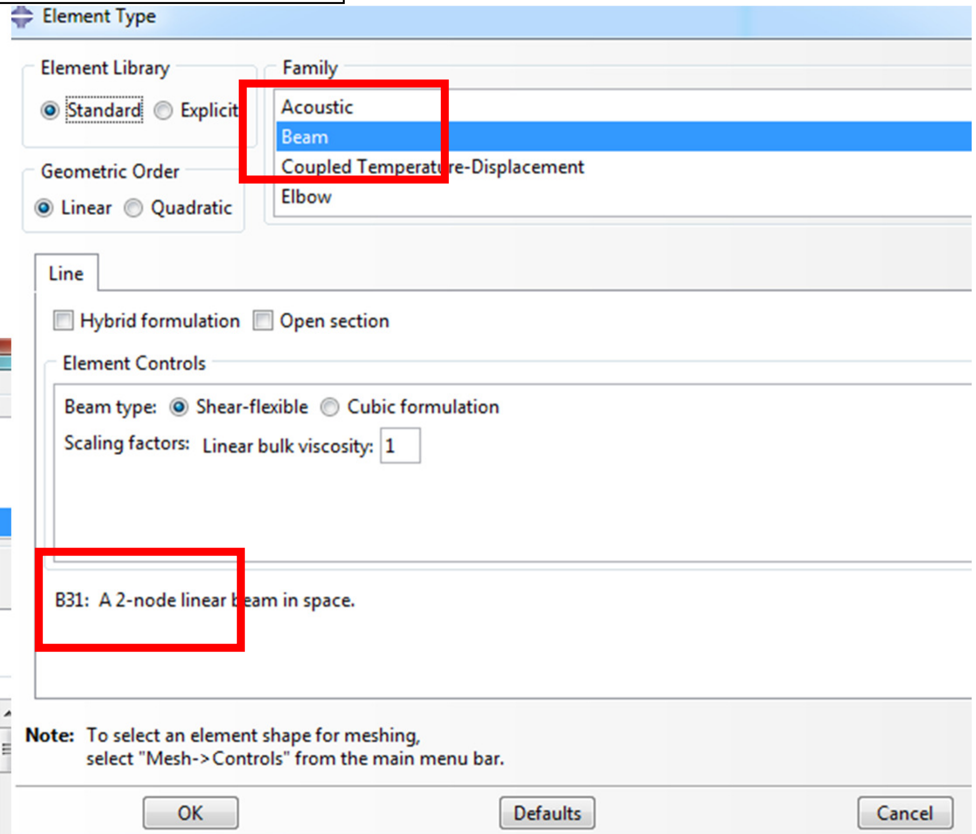


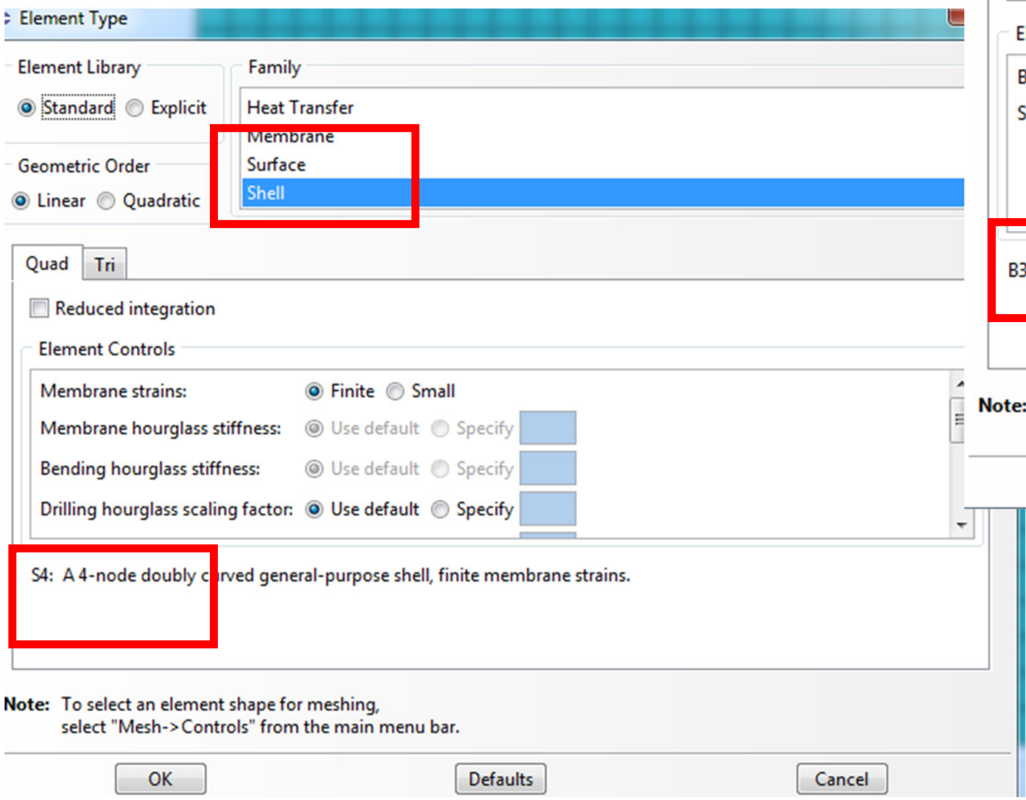
Numerical model

Structural analysis of composite slab

Mesh and finite elements:

Beams & → columns: B31 (0.3 m)





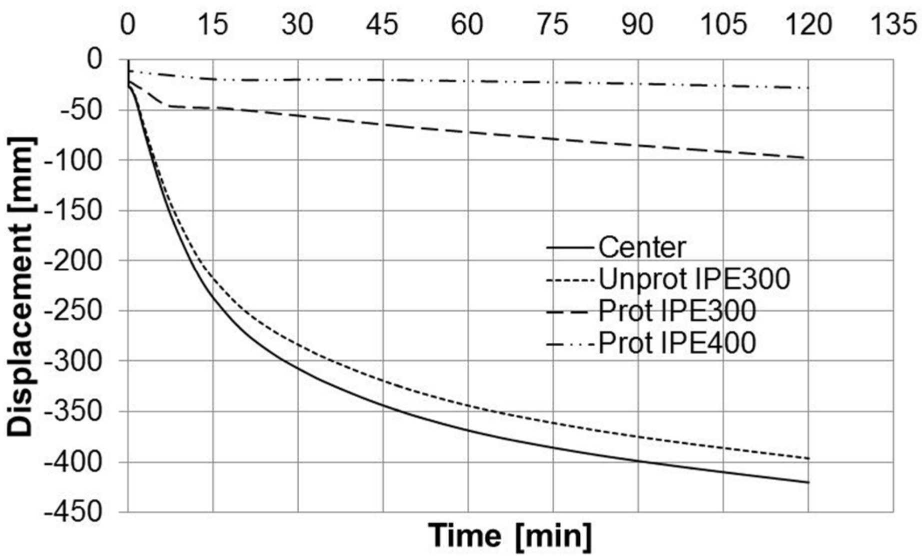
← Slab: S4 (0.3 m)

Numerical model - results

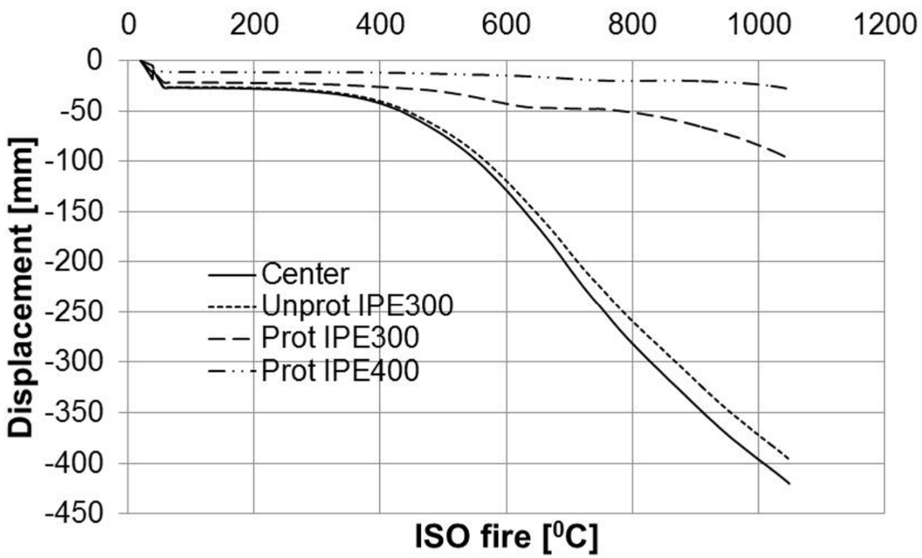
Structural analysis of composite slab

Results:

Displacements



With respect to time



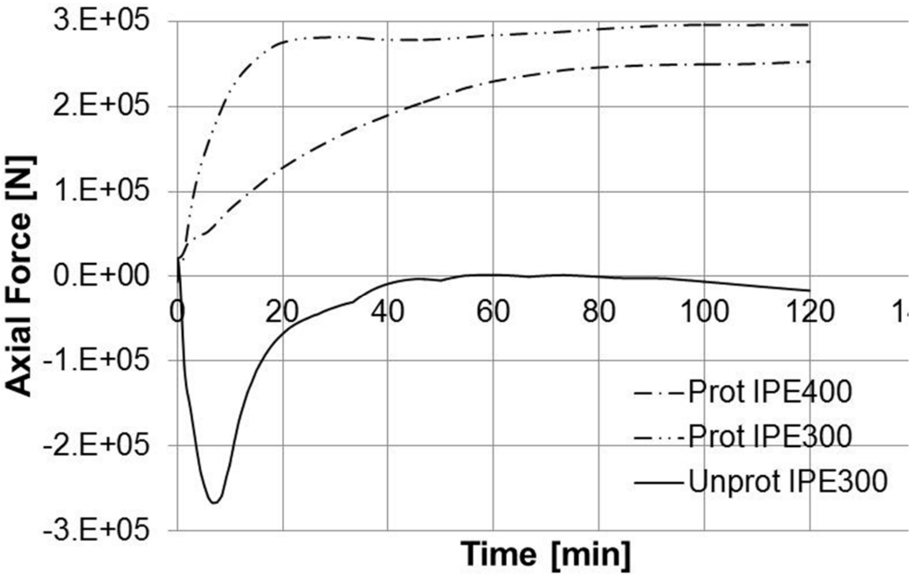
With respect to temperature

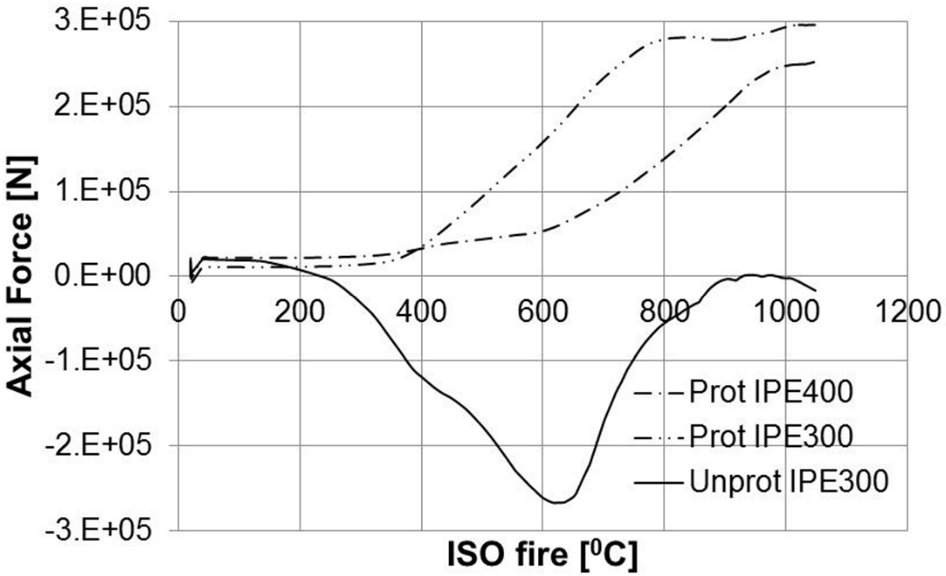
Numerical model - results

Structural analysis of composite slab

Results, other than obtained in experiment:

Axial force





With respect to time

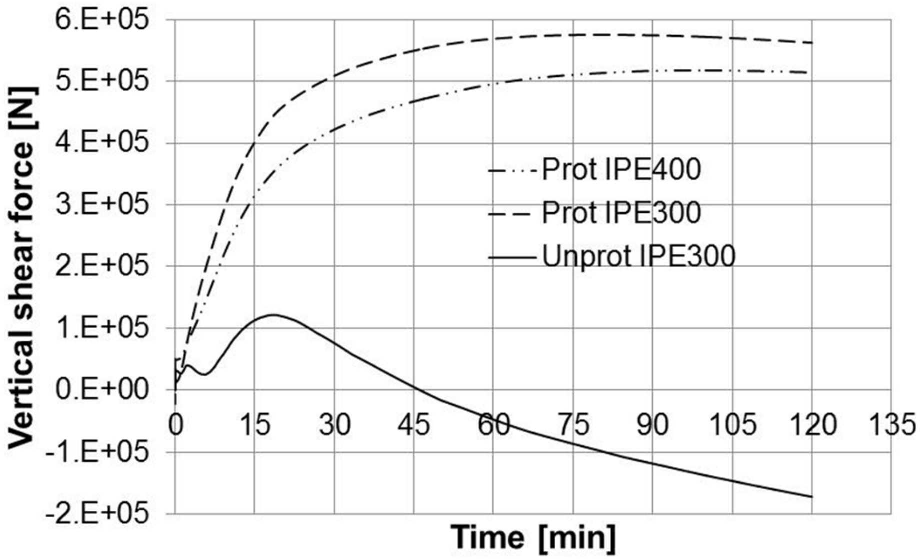
With respect to temperature

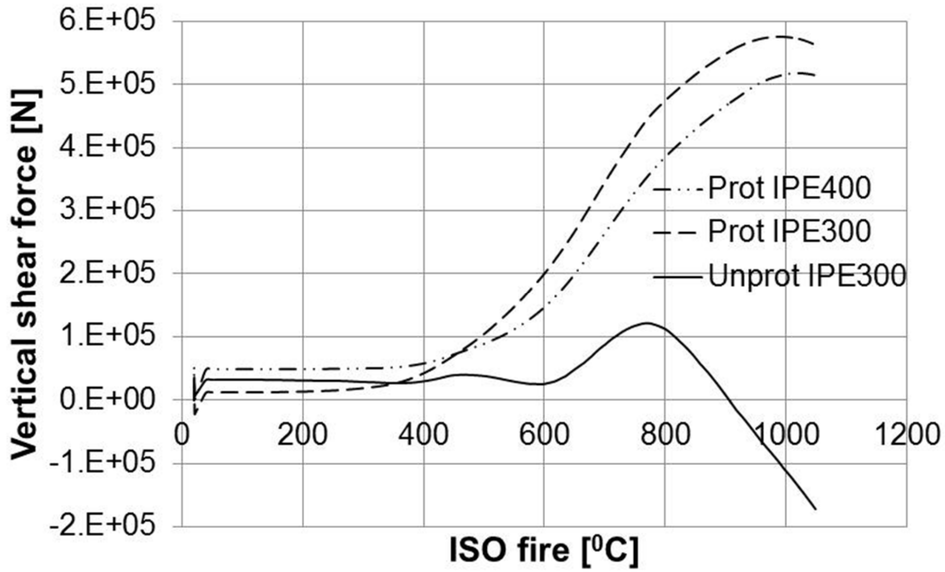
Numerical model - results

Structural analysis of composite slab

Results, other than obtained in experiment:

Vertical shear force





With respect to time

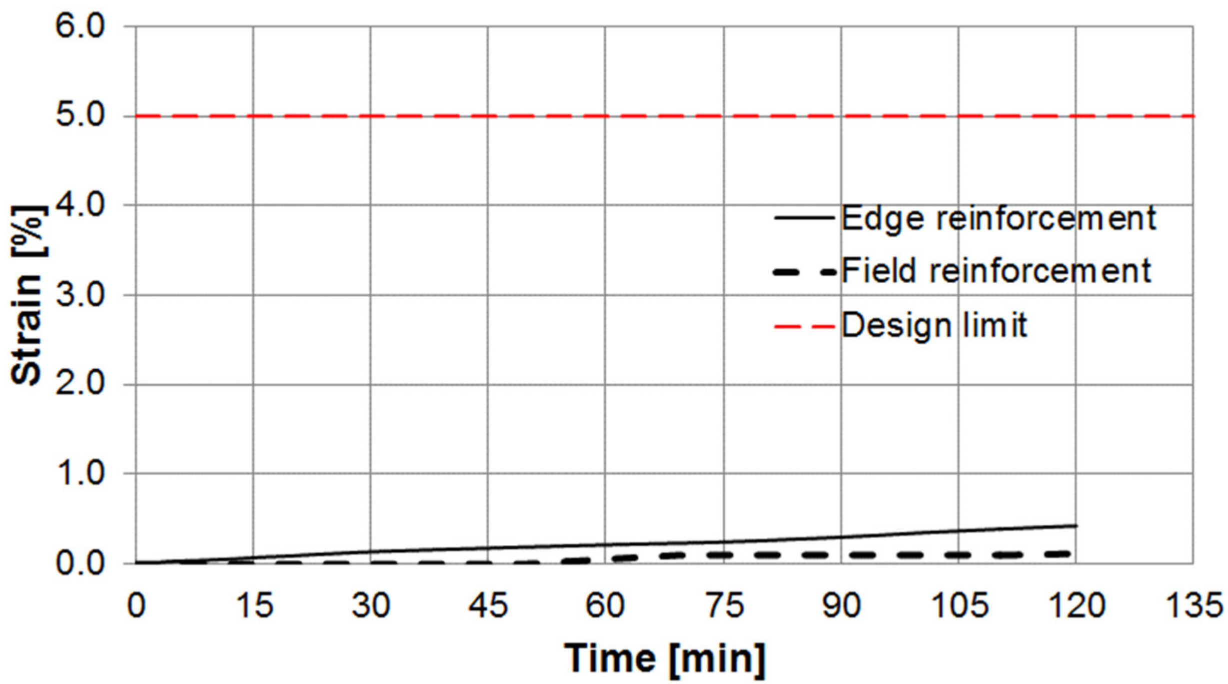
With respect to temperature

Numerical model - results

Structural analysis of composite slab

Results, other than obtained in experiment:

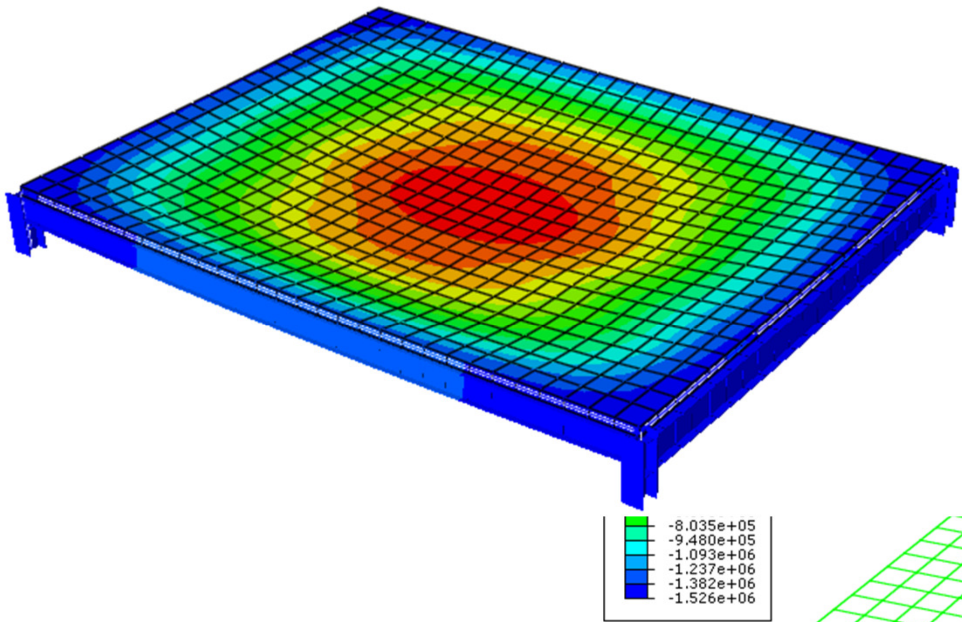
Reinforcement strain



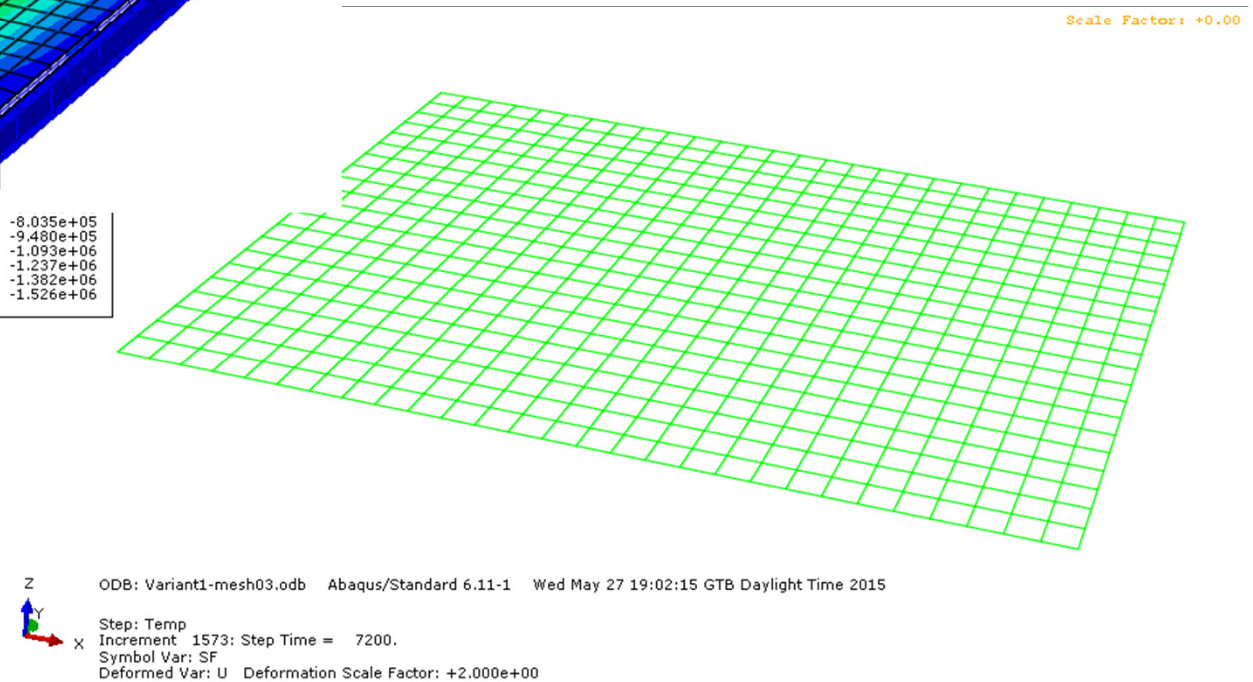
Numerical model - results

Structural analysis of composite slab

Deformed shape



Axial force



Numerical model - results

Mesh sensitivity study

Monitored variable: vertical displacement of the slab centre, U3

	Size [m]	Variable: U3 [cm]	Grid: h	Factor r	ε	Apparent order: p	Asympt solution	Extra-polated value	Approximate relative error	Extrapolated relative error	GCI_{12} [%]	GCI_{23} [%]
1	0.3	42.03	0.304	1.262	-0.27	4.160	42.20	42.20	0.0064	0.0039	0.49	0.84
2	0.4	41.76	0.383	-	-			-	-	-		
3	0.5	41.05	0.519	1.354	-0.71			42.04	0.0170	0.0067		

$$h = \left[\frac{1}{N} \sum_{i=1}^{N} \left(\Delta A_i \right) \right]^{1/2}$$

$$E_1 = \frac{\varepsilon}{r^p - 1}$$

$$\varepsilon = \frac{f_1 - f_2}{f_1}.$$

$$GCI = \frac{F_s\,|\varepsilon|}{r^p - 1} 100\%,$$

$$p = \frac{\ln\left(\dfrac{f_3 - f_2}{f_2 - f_1}\right)}{\ln(r)}$$

$$A_1 = \frac{f_1 - f_{h=0}}{f_{h=0}} \cong \frac{f_1 - f_2}{f_{h=0}} \frac{1}{r^p - 1},$$

h – representative value

N – number of elements

A_i – area of element “i”

r – refinement ratio (should have close values

ε – relative difference

A1 – relative error

p- order of convergence

GCI – grid convergence index

Conclusions of composite slab benchmark

- **A benchmark represents a step-by-step procedure for a thermo-mechanical analysis in a certain FEM computer program, here Abaqus;**

- **The procedure used in Abaqus was validated with the experimental results from the fire test, by means of the same numerical model, for which a complete input data was considered.**

Verification example

- By comparison to existing experimental tests

- German annex EN1991-1-2

  - Comparison examples

There are 11 examples analyzing :

- heat transfer for different material properties
- elongation of elements
- forces and stresses
- fire resistance time

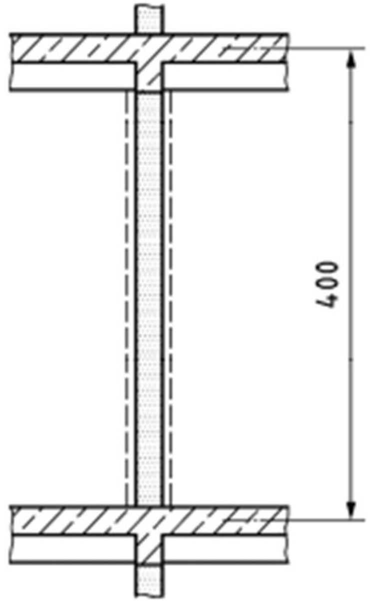
For each example there are given results and their error tolerances

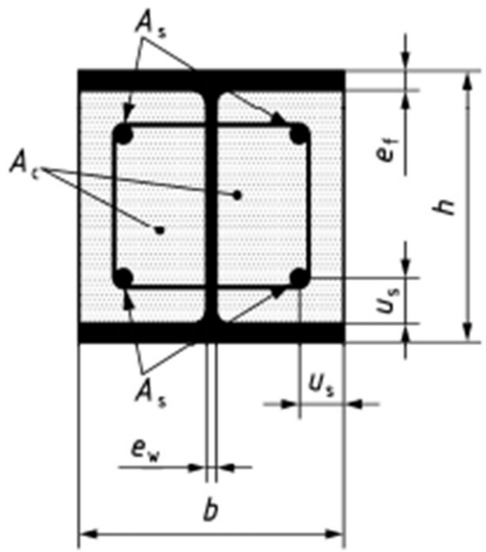
Example 11 - DIN1991-1-2

Parabolic longitudinal path – midspan imperfection L/1000

Dimensions		l / h / b	cm	400 / 30 / 30
		u_s	mm	50
		e_f	mm	19
		e_w	mm	11
Buckling length		$l_{0,fi}$	cm	200
Load		$N_{E,fi,d,t}$	KN	-1700
Concrete C25/30 (3 % moisture (by mass))		$f_{ck(20°C)}$	N/mm²	25
Reinforcing steel S 500		$f_{yk(20°C)}$	N/mm²	500
Structural steel S 235		$f_{ak(20°C)}$	N/mm²	235
Stress – strain curve		Concrete a.		DIN EN 1994-1-2
		Reinforcing steel b.		
		Structural steel		
Temperature load		ETK (four sides)		DIN EN 1991-1-2
Heat transfer coefficient		α_c	W/(m² × K)	25
Emissivity		ε_m		0.7
Thermal and physical material properties	Concrete	λ, ρ, c_p, $\varepsilon_{th,c}$		DIN EN 1994-1-2
	Steel	λ, ρ_s, c_a, $\varepsilon_{th,s}$, $\varepsilon_{th,a}$		DIN EN 1994-1-2

a. Containing mainly quartzite aggregate and density ρ=2400 kg/m³
b. Hot-rolled



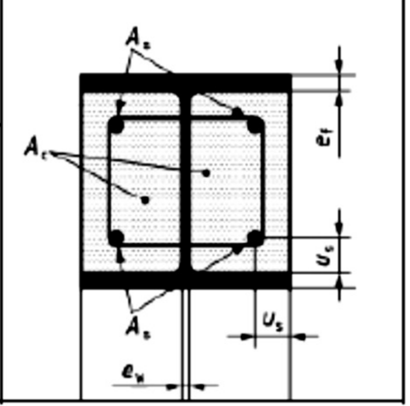


Analysis

Previous verification using SAFIR

- 2D beam elements
- For displacement at 60 minutes the criterion is not fulfilled

Requested results		Reference value	Calculated value	Deviation [%]	Limit [%]
Failure time		92	88	−4.35	± 5
Displ. [mm]	30 min	4.40	4.44	+0.82	
	60 min	5.50	5.04	**−8.18**	



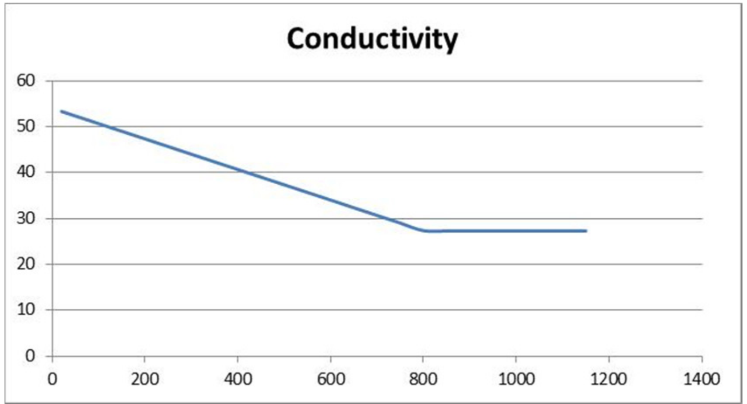
The result is not consistent with the other values:

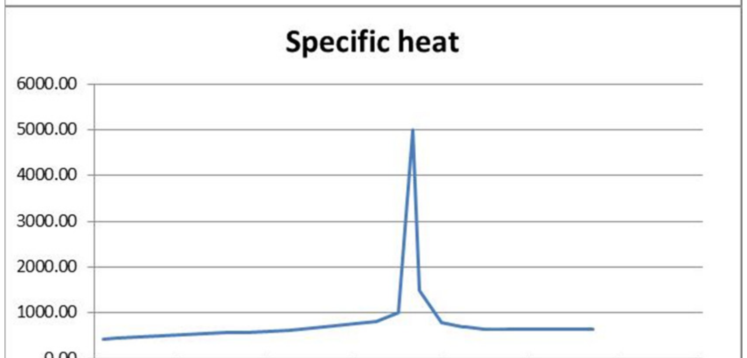
- at 30min the displacement is bigger than the reference value while for 60 minutes the displacement is smaller than the reference value
- For smaller displacement it is expected longer resistance time; but for this case the fire resistance is smaller

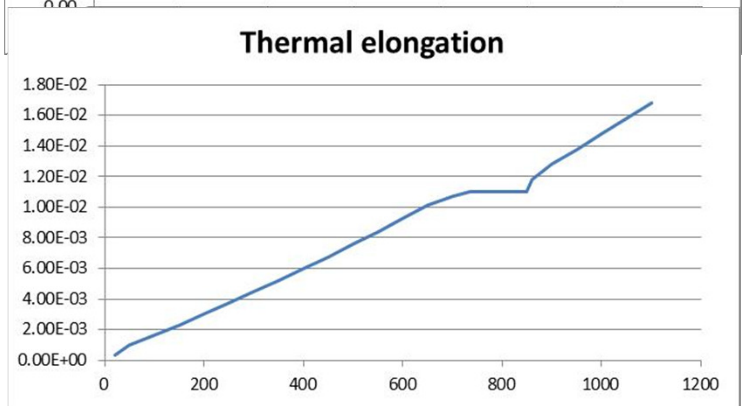
Model - Abaqus

Material properties

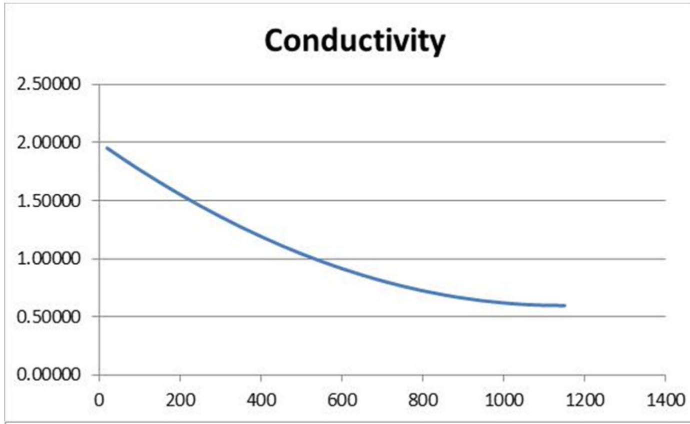
Steel

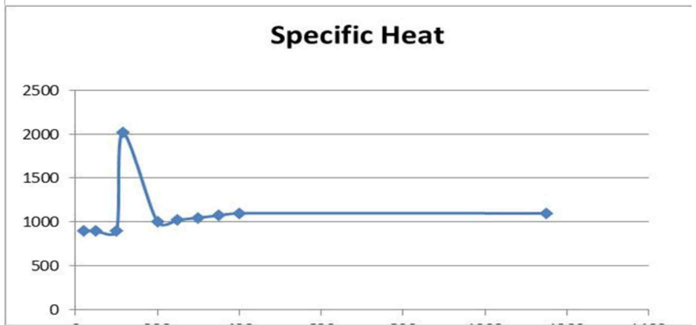


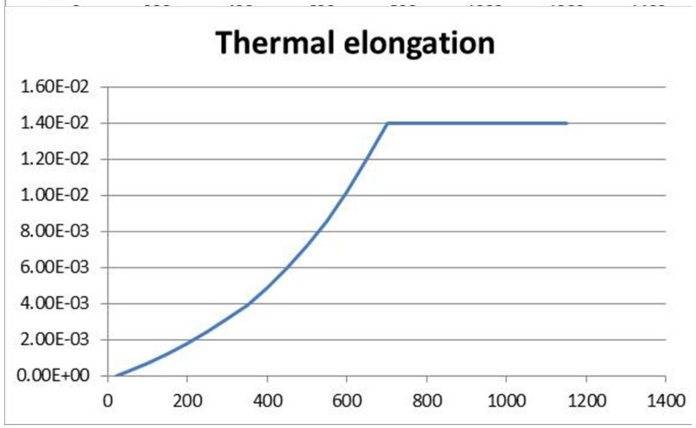




Concrete





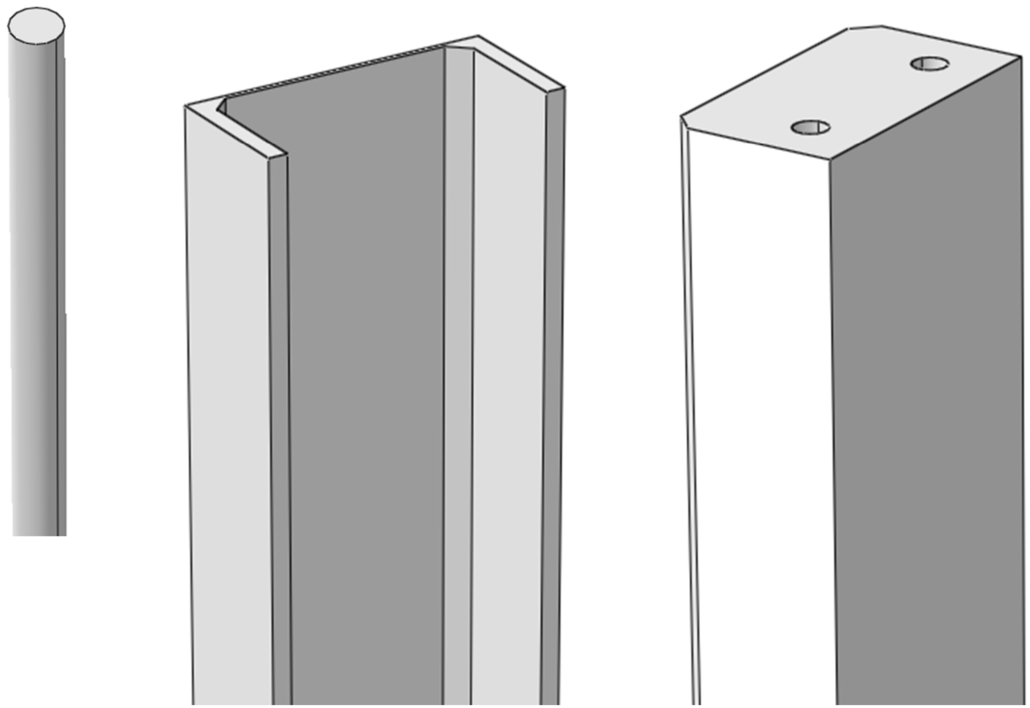


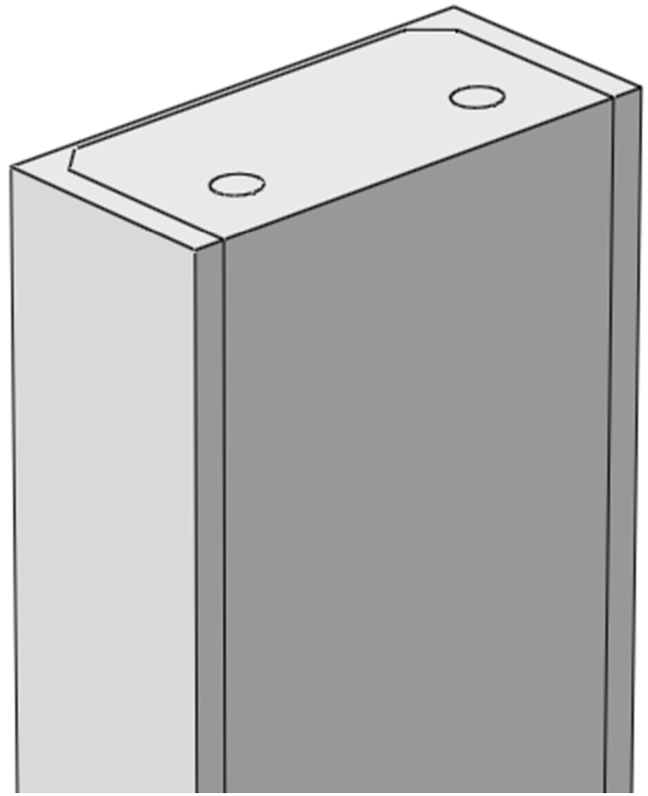
Model

Part definition

Separated parts for
each material
-contact problem

One part divided
into partitions





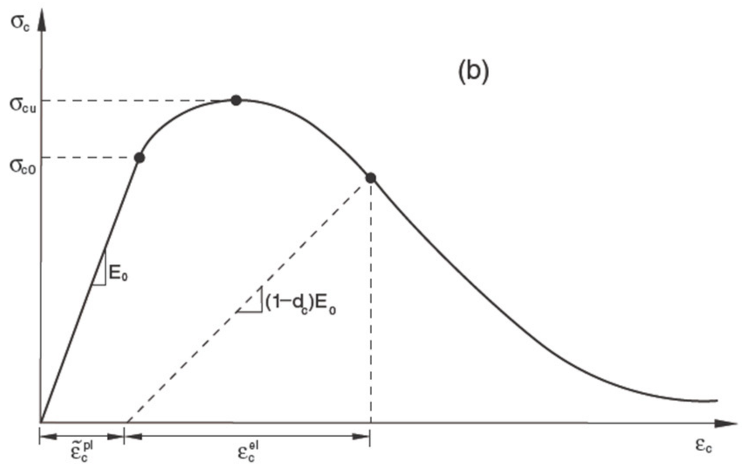
Model

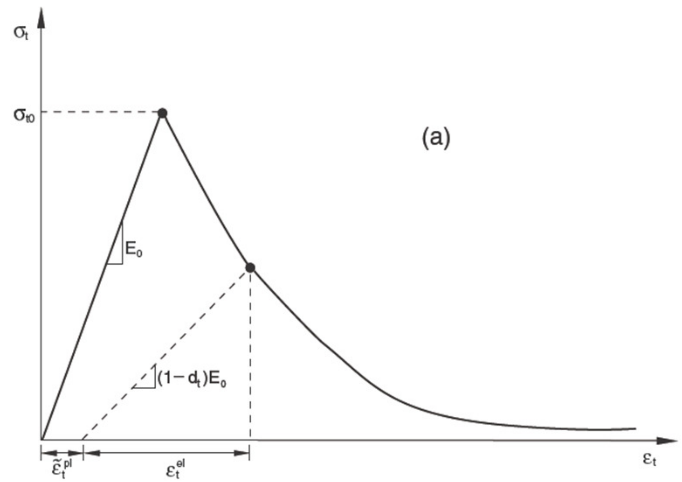
Concrete model

Piecewise stress-strain → Large tensile stress for concrete

Smeared concrete → -elongation & BC problems

Damaged plasticity concrete



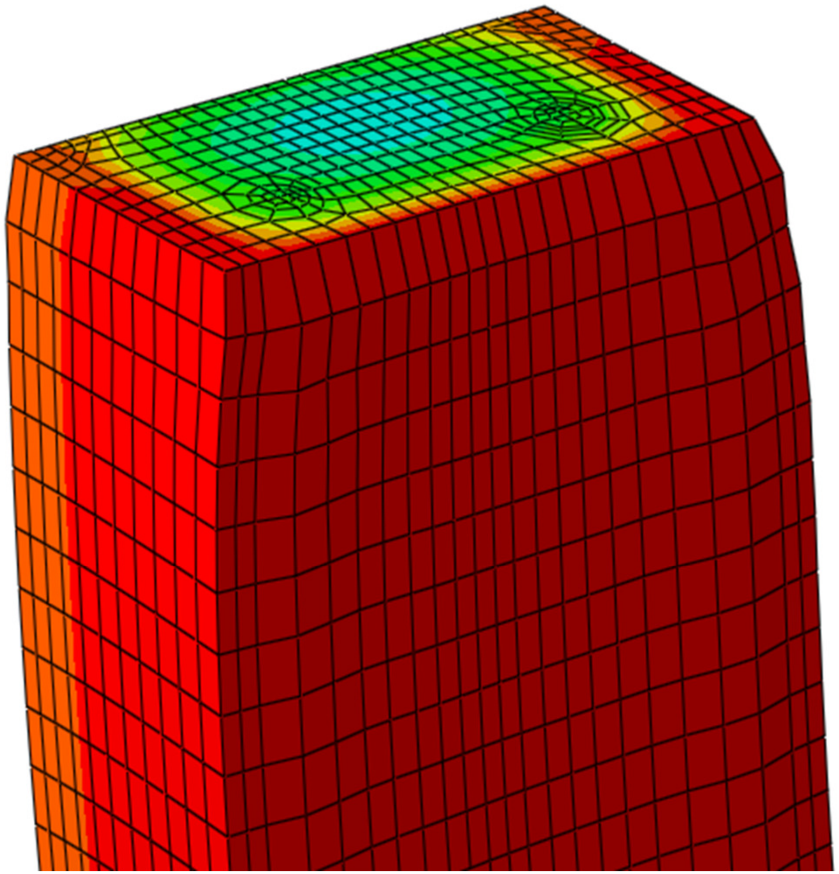


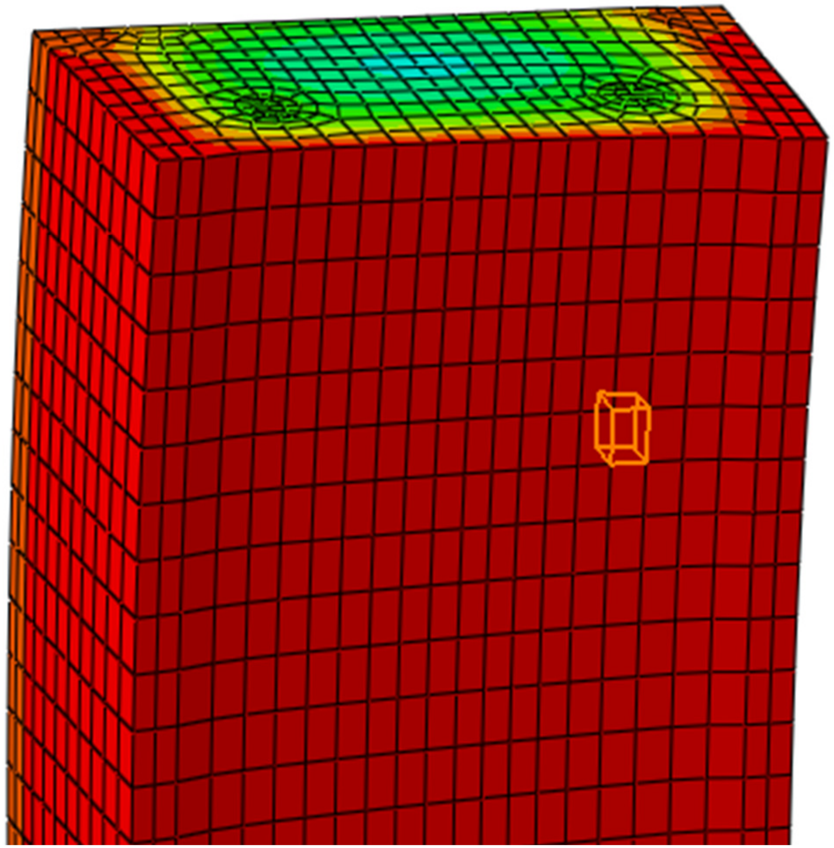
Model

Boundary conditions

Fully fixed
-surface

Fully fixed
-node



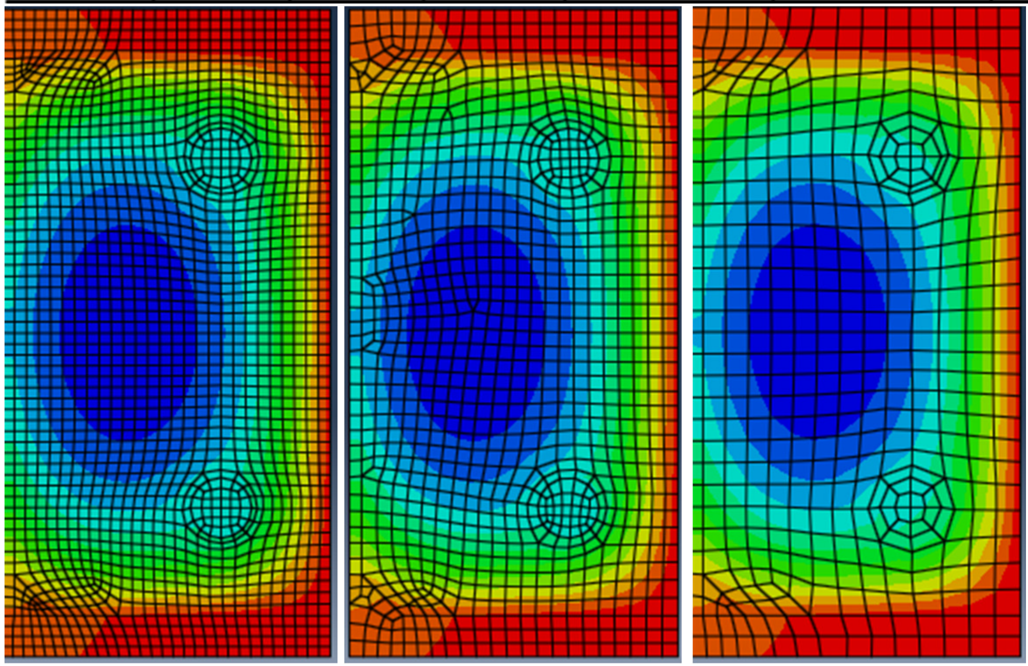


Analysis

Thermal analysis - 2D - verification

Grid Convergence Index (GCI) for temperatures at 90min

ABAQUS (2D)			Temp	refinement ratio (r)	Order of convergence HEB (p)	Asymptotic solution $f_{h=0}$	ε	GCI_{12} [%]	GCI_{23} [%]	$GCI_{23}/r^{p}GCI_{12}$
	mesh	size	HEB							
	1	0.0025	451.00				-0.0011086			
	2	0.005	451.50	1.414213562	0.844466001	449.5294118		-0.4076	-0.54557	0.998893
Thermal	3	0.01	452.17				-0.0014839			



[redacted]	Abaqus	Safir		DIN
HEB300	452	459	>	447
[redacted]	512	518	<	535

Analysis

Structural analysis

Sequentially coupled analysis

Thermal analysis

- 3D solid – DC3D8: An 8-node linear heat transfer brick

Mechanical analysis

- 3D solid – C3D8R: An 8-node linear brick, reduced integration, hourglass control

Fully coupled analysis

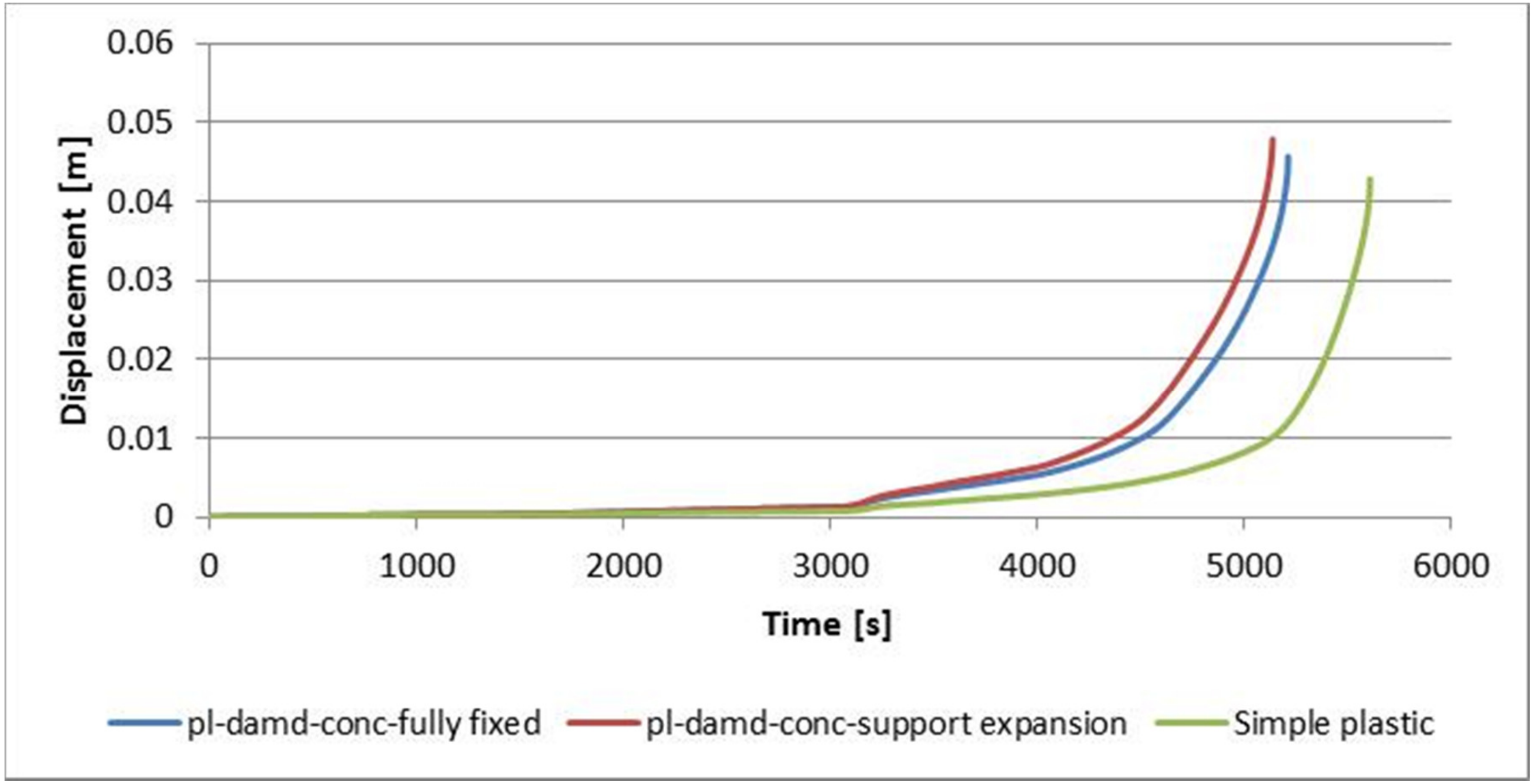
Thermal and mechanical analysis

- 3D solid – C3D8T

An 8-node thermally coupled brick, trilinear displacement and temperature

Results

Fully coupled analysis



Results

The results do not fulfill 2 criterion: fire resistance time and displacemen

ABAQUS		Reference value X	Calculated value X′	Deviation [%]	Limit [%]
Failure time		92	87	-5.43	± 5
Displ. [mm]	30 min	4.40	4.56	+3.52	
	60 min	5.50	7.82	+42.16	

It can be observed that, as is case of SAFIR analysis, fire resistance time is smaller than the reference value. On the other hand, displacement for 60 minutes is higher than the reference value (still, the Abaqus results are consistent since for a smaller resistance time there is a corresponding larger deflection for the column, with respect to the reference value).

SAFIR		Reference value	Calculated value	Deviation [%]	Limit [%]
Failure time		92	88	−4.35	± 5
Displ. [mm]	30 min	4.40	4.44	+0.82	
	60 min	5.50	5.04	−8.18	

Conclusions for composite column

- Extra information needed for the calculation model
- Mechanical interaction between elements - unknown
- Concrete model - difficult to define
- Analysis time – long
- What happens after debonding? –thermal condition

Bibliography

[1] R. Zaharia, C. Vulcu, O. Vassart, T. Gernay and J.M. Franssen, "Numerical analysis of partially fire protected composite slabs," *Steel and Composite Structures*, vol. 14, no. 1, pp. 21-39, 2013

[2] Florian Block, Fire Engineering in Practice – State of the Art in Performance-based Design, *COST TU904 – 2013 Training School – Naples 7th of June 2013*

[3] Fire Behaviour of Steel and Composite Floor Systems – FRACOF - http://www.macsfire.eu/uk-start.html

[4] Abaqus, Documentation Abaqus 6.11

[5] Dictionary.com

[6] Wald, F., Burgess, I., Kwasniewski, L., Horova, K. and Caldova, E. (ed.). 2014. *Benchmark studies-Verification of numerical models in fire engineering*, Prague: CTU Publishing House

[7]Bin Zhao, Mohsen Roosefid, and Olivier Vassart, "Full scale test of a steel and concrete composite floor exposed to ISO fire," in Structures in fire (Proceedings of the Fifth International Conference), Singapore, 2008, pp. 539-550

EN 1994-1-2 Eurocode 4 - Design of composite structures - Part 1-2: General rules - Structural fire design

Ioan BOTH, PhD. Eng.

ioan.both@upt.ro