



REPUBLIKA SLOVENIJA
MINISTRSTVO ZA IZOBRAŽEVANJE,
ZNANOST IN ŠPORT

Univerza v Ljubljani



EVROPSKA UNIJA
EVROPSKI
SOCIALNI SKLAD
NALOŽBA V VAŠO PRIHODNOST

STRUCTURAL FIRE DESIGN

Considerations in fire design of steel structures

Ioan Both, PhD. Eng

Politehnica University of Timisoara
Department of Steel Structures and Structural Mechanics

Ljubljana, 24 Oct - 3 Nov 2017

Naložbo sofinancirata Evropska unija iz Evropskega socialnega sklada in Republika Slovenija.

Context. Why fire design?

Materials

Actions on structures

Basic design considerations

Fire design considerations

Examples

Context. Why?

- Source of energy and heat

Fire effect 1



Fire effect 2



- The cause of material and life loss

Fire effect 3



Fire effect 4



Fire effect 5



Fire effect 6

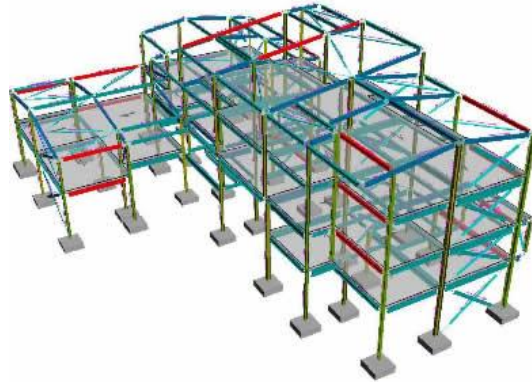


Context

Architects



Structural Engineers

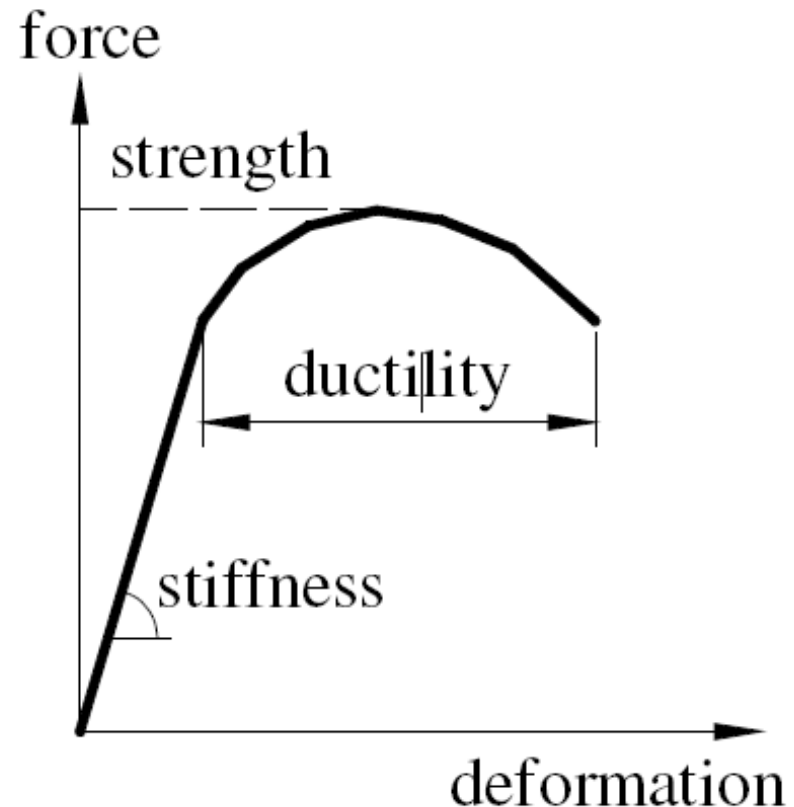


Service Engineers



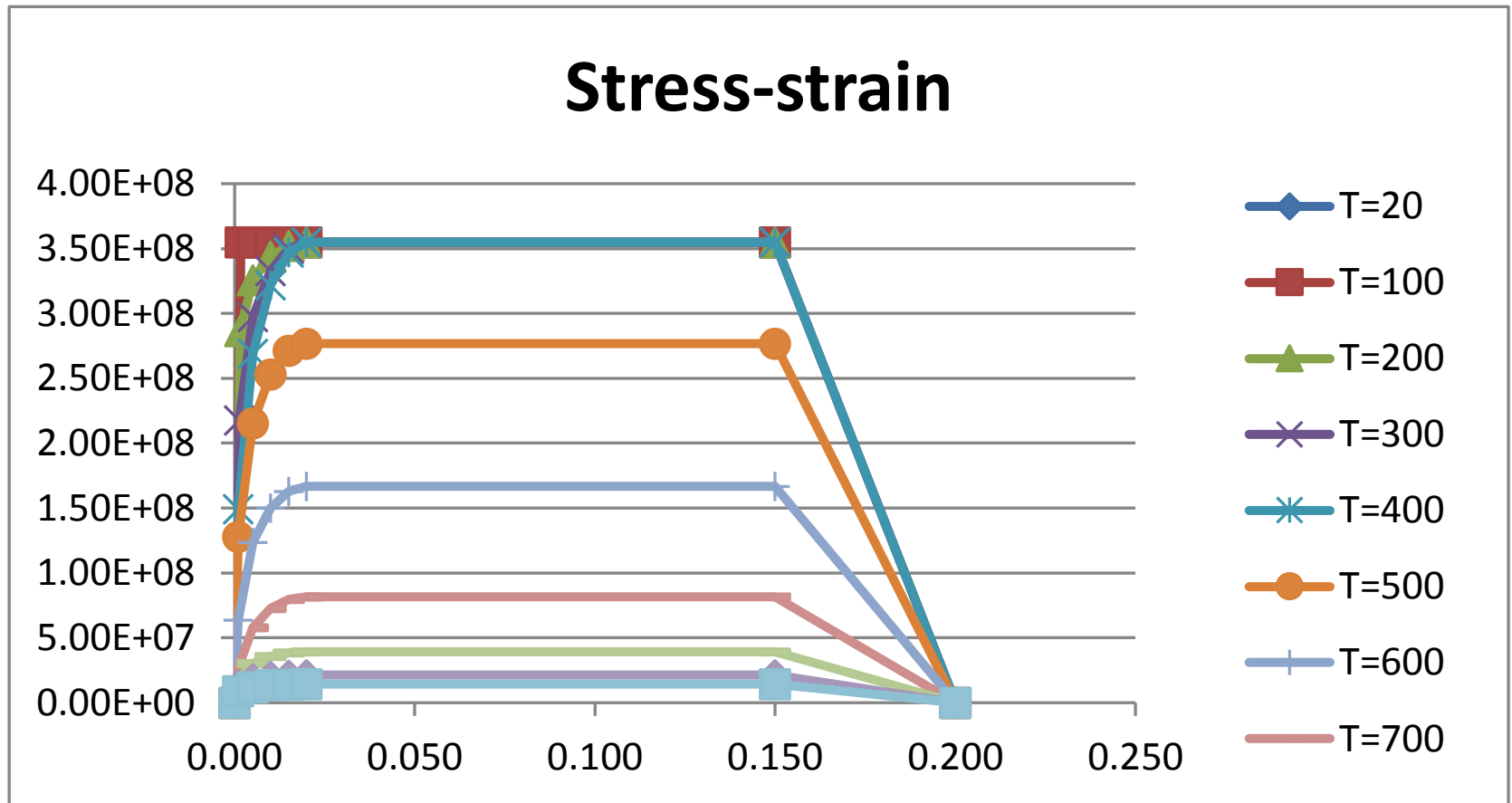
Structural materials

- Structural material is the one which is used in those parts of the structure which carry loads and give it strength and stiffness
- Properties of structural materials:
 - strength
 - stiffness
 - ductility



Structural materials: ductility

Ductile materials: able to deform significantly into the inelastic range



Design considerations

Actions are classified by their variation in time as follows:

- permanent actions (G), e.g. self-weight of structures, fixed equipment and road surfacing, and indirect actions caused by shrinkage and uneven settlements;
- variable actions (Q), e.g. imposed loads on building floors, beams and roofs, wind actions or snow loads;
- accidental actions (A), e.g. explosions, or impact from vehicles

FIRE

Several loads → load combinations

Actions on structures

Based on codes

EUROPEAN STANDARD	EN 1990:2002+A1
EUROPEAN STANDARD	EN 1991-1-1
EUROPEAN STANDARD	EN 1991-1-3
EUROPEAN STANDARD	EN 1991-1-4
NORME EUROPÉENNE	EN 1991-1-5
EUROPÄISCHE NORM	
EUROPEAN STANDARD	Novembre 2003

General actions

Snow

Wind

Temperature

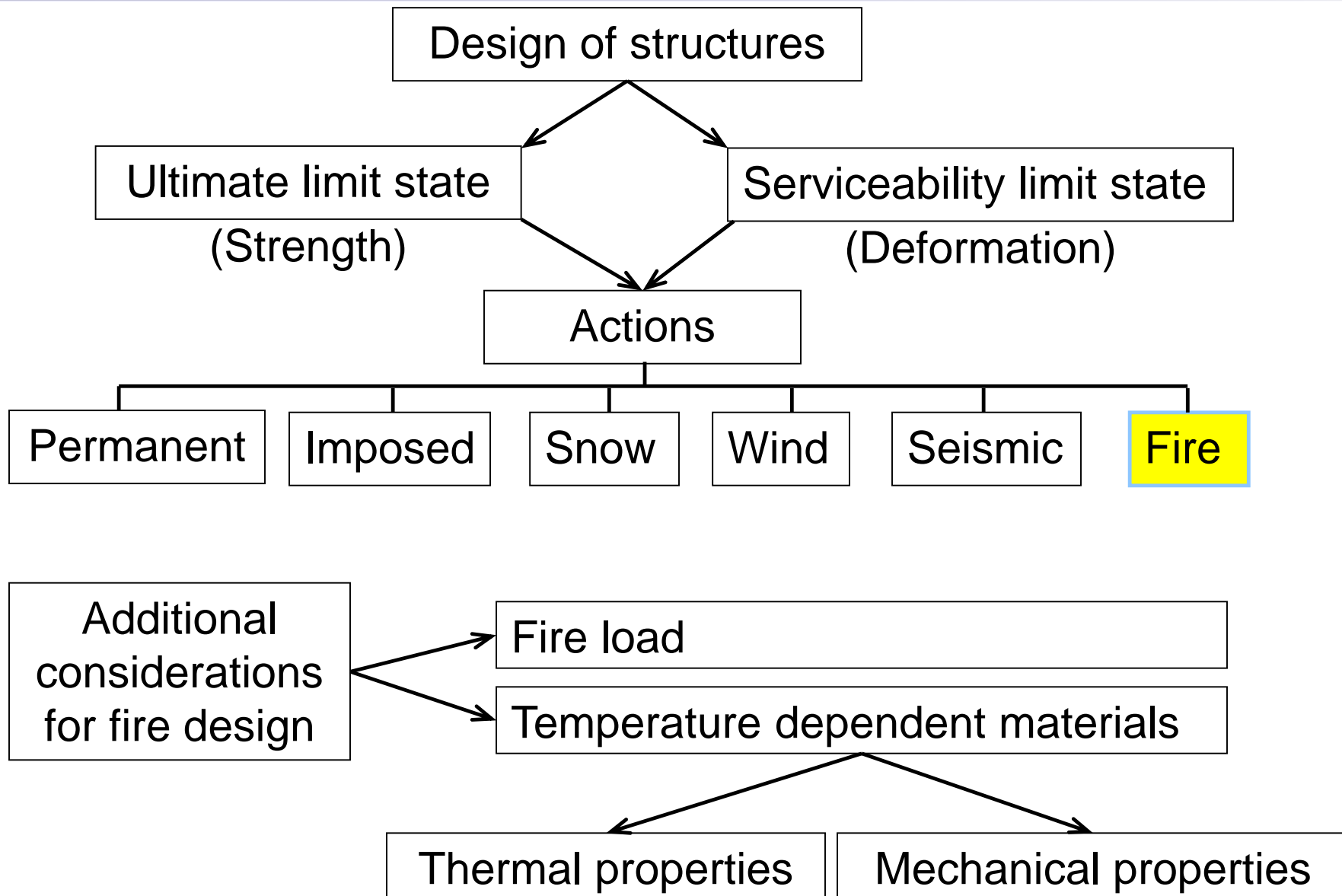
Exterior
Wall

ICS 91.010.30

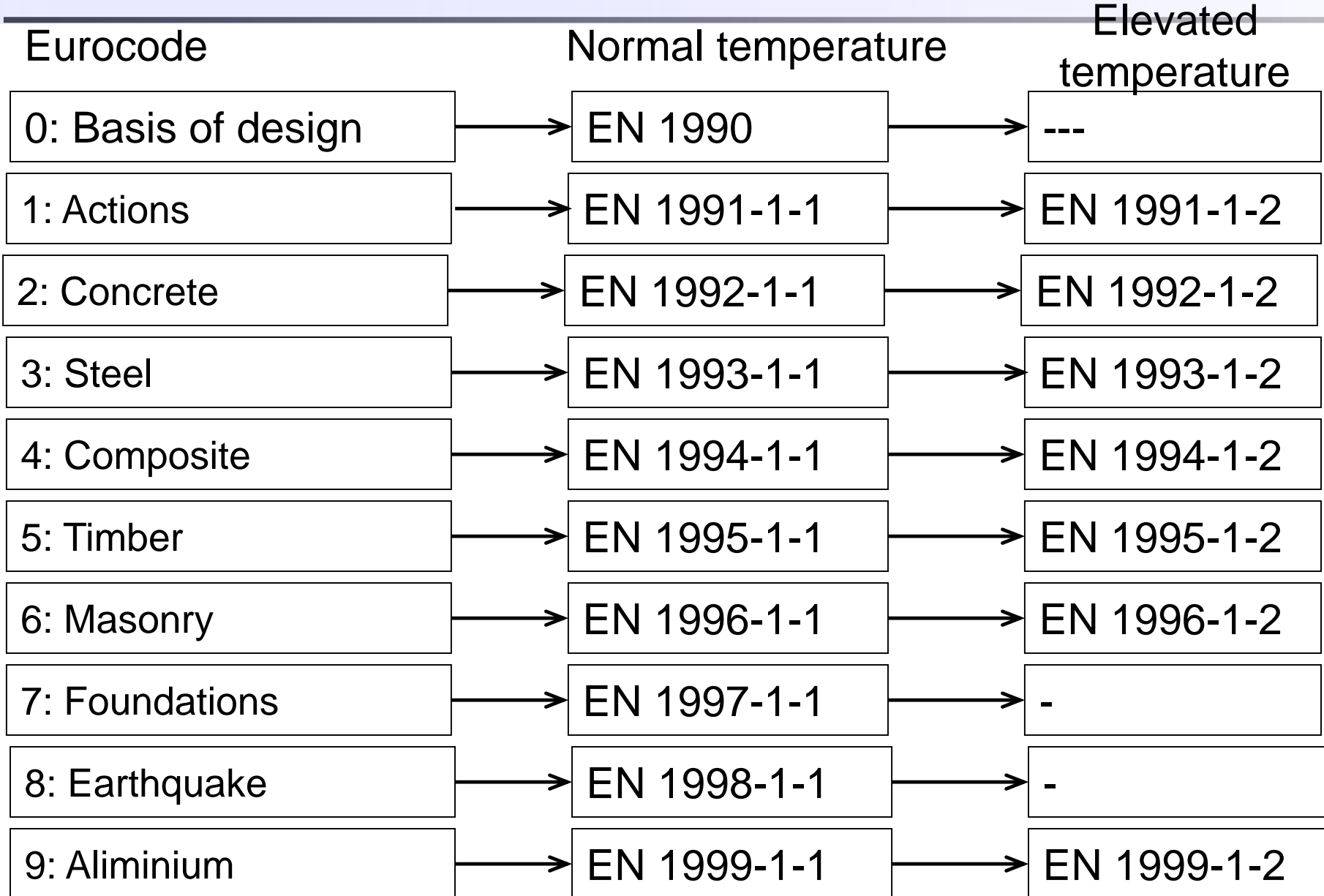
Version Française

Eurocode 1: - Actions sur les structures - Partie 1-5: Actions
générales - Actions thermiques

Design considerations

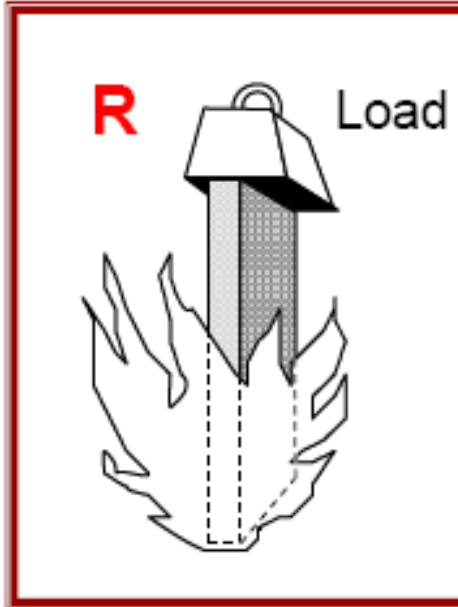


Fire action in design codes



Design considerations

Resista



Design considerations

The aim of fire design codes

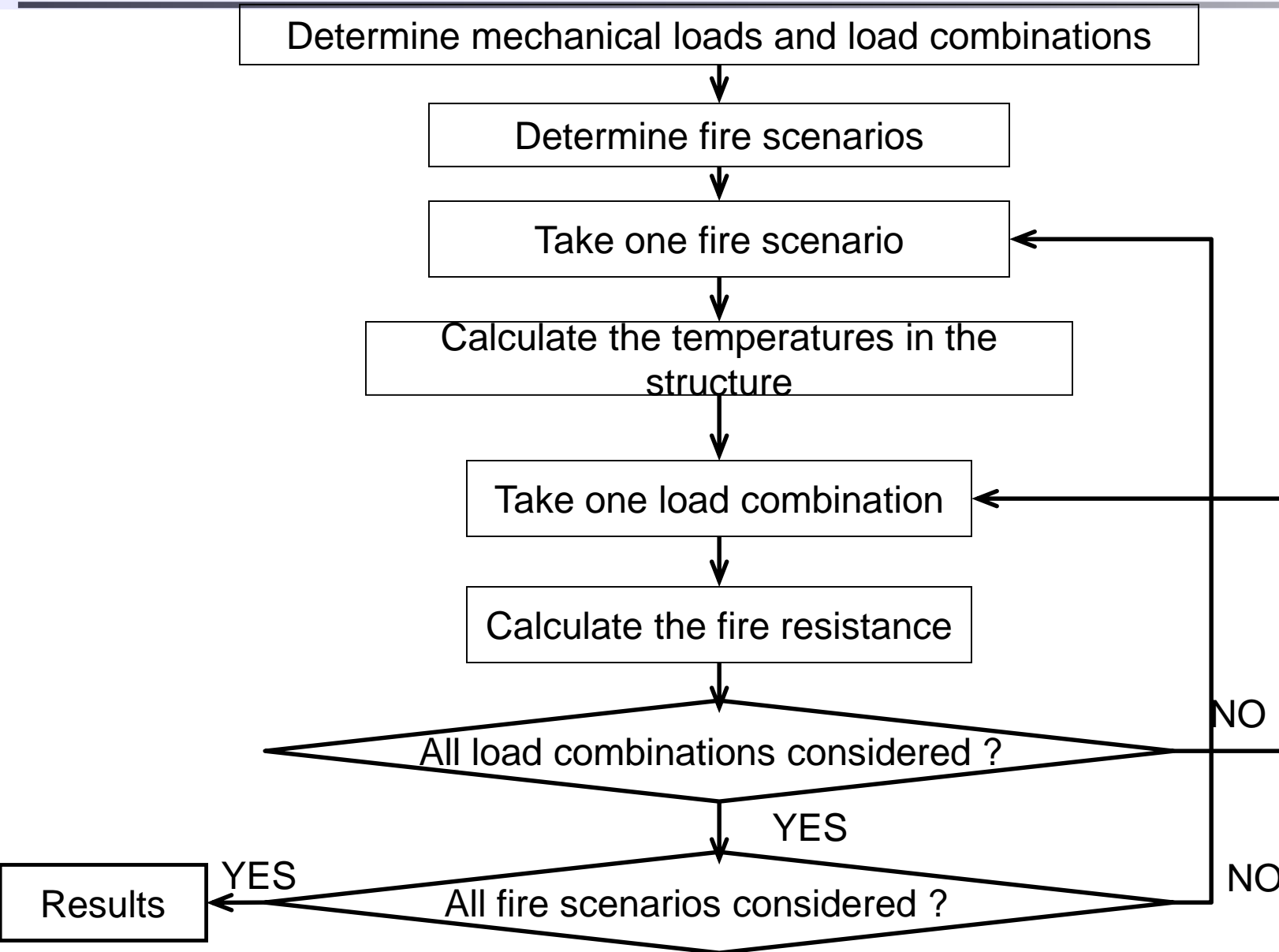
- The Eurocodes covers the Resistance criterion - the load bearing capacity of elements and structures. It gives information that allows calculating whether or how long a structure is able to withstand the applied loads during a fire. The design is performed in the ultimate limit state.
- The EN fire design codes **DO NOT** relate to the insulation or integrity criteria of structural elements (E or I).

The aim of fire design codes

- There is no deformation criteria explicitly mentioned in the Eurocode.
- Deformation criteria should be applied in two cases:
 1. When the fire protection may lose its efficiency in case of excessive deformations.
 2. When separating elements, for example a separating wall, supported by or located under a structural element may suffer from excessive deformations of this member

Fire design considerations

Steps of an analysis



Fire design considerations

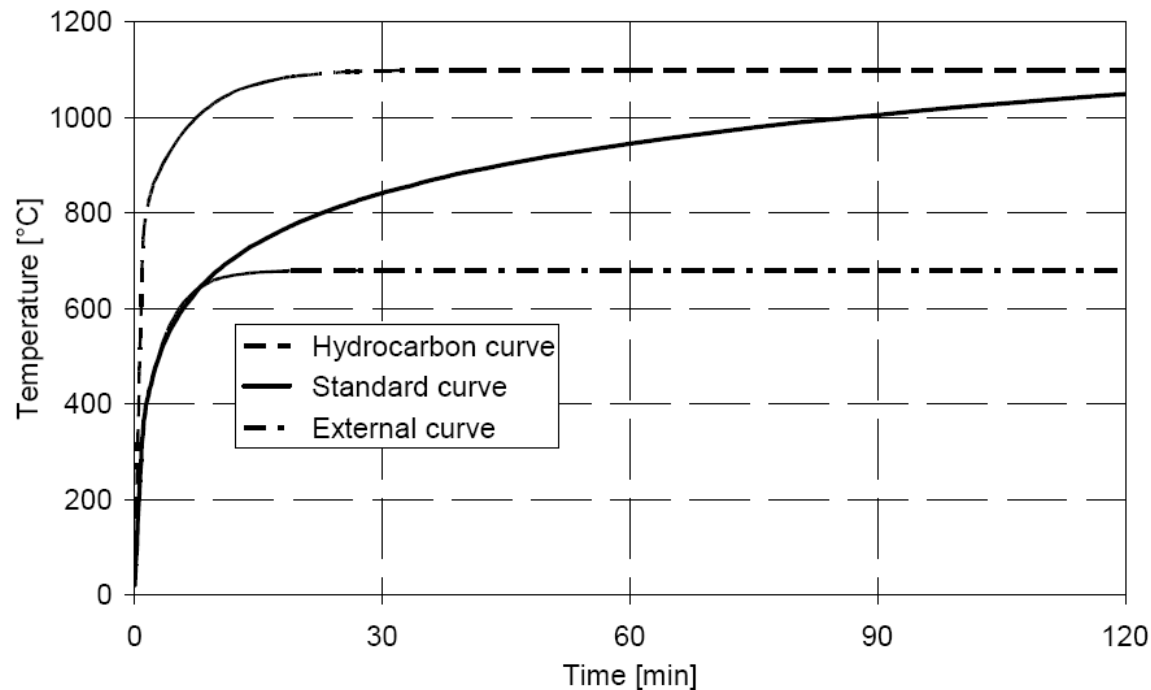
Thermal action

the fire is represented by a temperature-time curve, i.e. an equation describing the evolution with time of the unique temperature that is supposed to represent the environment in which the structure is located.

1. Standard fire (ISO 834)

2. External curve

3. Hydrocarbon curve



Fire design considerations

Mechanical actions

The design against the ultimate limit state is based on the comparison between the **RESISTANCE OF THE STRUCTURE** calculated with the design values of material properties, and the **EFFECTS OF ACTIONS** calculated with design value of actions.

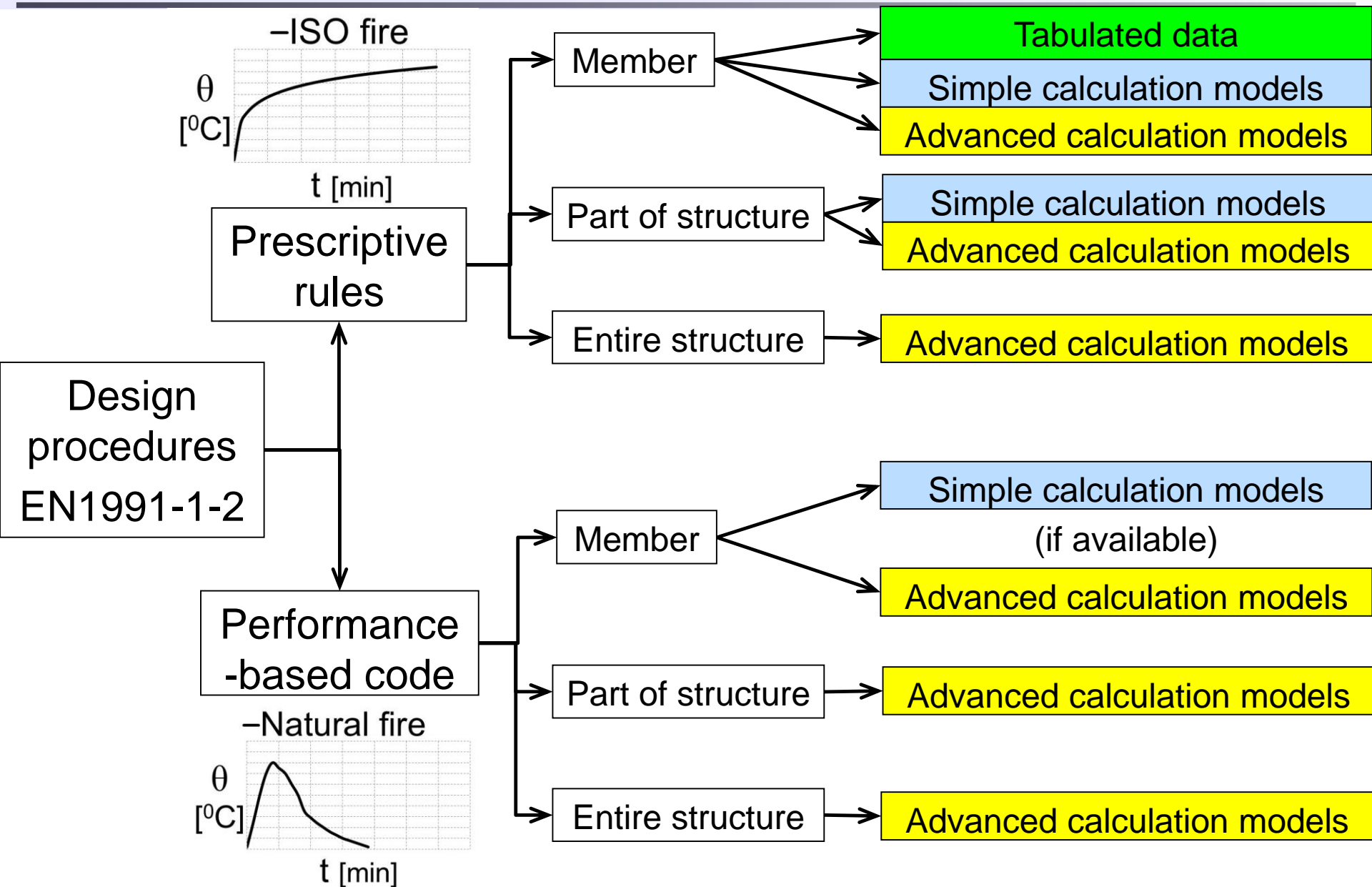
$$R_{fi,d,t}(X_{d,fi}) > E_{fi,d}(F_{fi,d})$$

$$E_{fi,d} = G_k + P_k + \psi_{1,1} Q_{k1} + \sum_{i>1} \psi_{2,i} Q_{ki}$$

$$E_{fi,d} = G_k + P_k + \sum_{i=1} \psi_{2,i} Q_{ki}$$

Action	ψ_1	ψ_2
Imposed load in buildings		
category A: domestic, residential	0.5	0.3
category B: offices	0.5	0.3
category C: congregation areas	0.7	0.6
category D: shopping	0.7	0.6
category E: storage	0.9	0.8
Traffic loads in buildings		
category F: vehicle weight $\leq 30\text{kN}$	0.7	0.6
category G: $30\text{kN} < \text{vehicle weight} < 160\text{kN}$	0.5	0.3
category H: roofs	0.0	0.0
Snow loads		
for sites located at altitude $H \leq 1000\text{ m}$	0.2	0.0
for sites located at altitude $H > 1000\text{ m}$	0.5	0.2
Wind loads	0.5	0.0

Fire design considerations



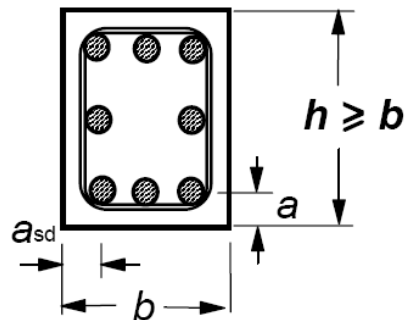
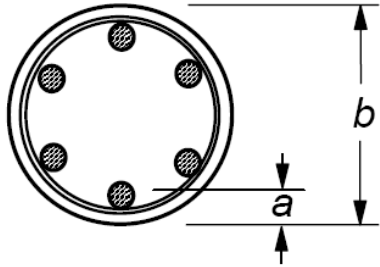
Fire design considerations

Tabulated data:

recognised design solutions for the standard fire exposure

Columns

$$\mu_{fi} = N_{Ed,fi} / N_{Rd}$$



Standard fire resistance	Minimum dimensions (mm) Column width b_{min} /axis distance a of the main bars			
	Column exposed on more than one side			Exposed on one side
	$\mu_{fi} = 0.2$	$\mu_{fi} = 0.5$	$\mu_{fi} = 0.7$	$\mu_{fi} = 0.7$
1	2	3	4	5
R 30	200/25	200/25	200/32 300/27	155/25
R 60	200/25	200/36 300/31	250/46 350/40	155/25
R 90	200/31 300/25	300/45 400/38	350/53 450/40**	155/25
R 120	250/40 350/35	350/45** 450/40**	350/57** 450/51**	175/35
R 180	350/45**	350/63**	450/70**	230/55
R 240	350/61**	450/75**	-	295/70

**

Minimum 8 bars

For prestressed columns the increase of axis distance according to 4.2.2. (4) should be noted.

Fire design considerations

Simple calculation models:

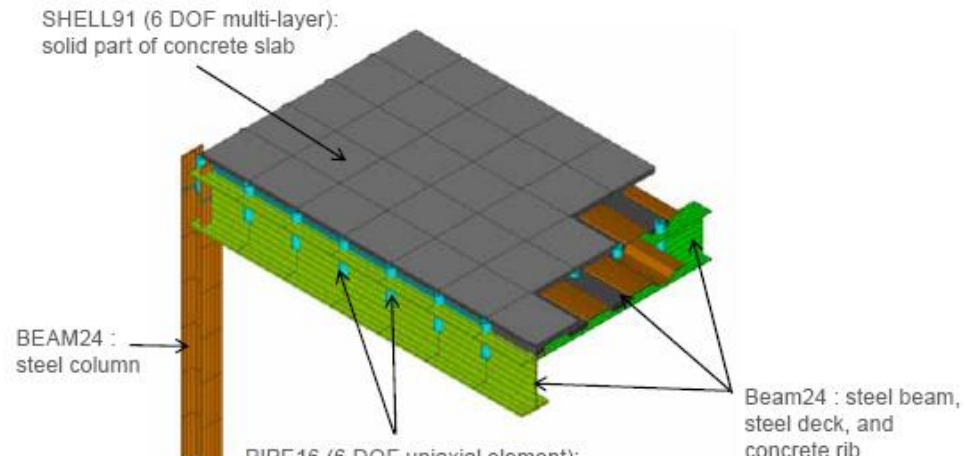
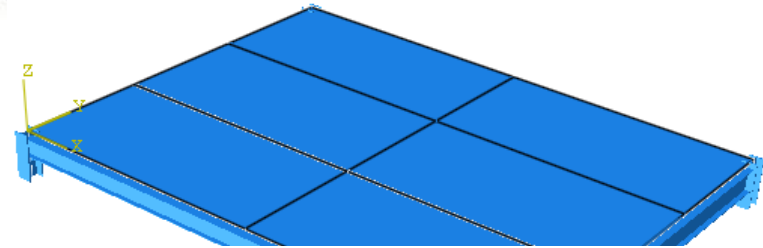
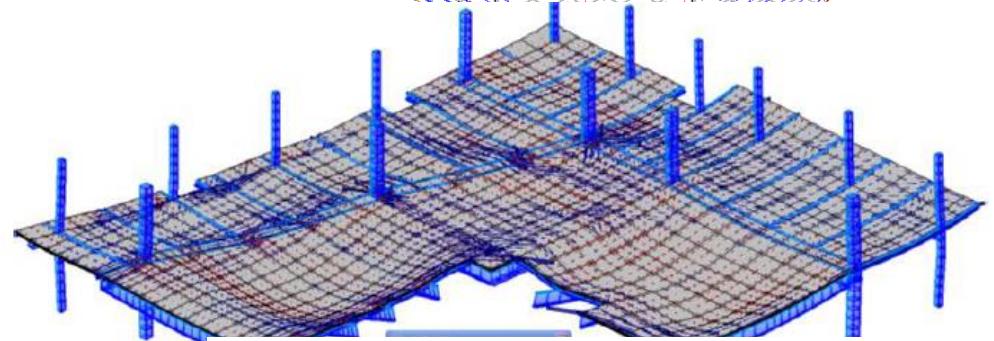
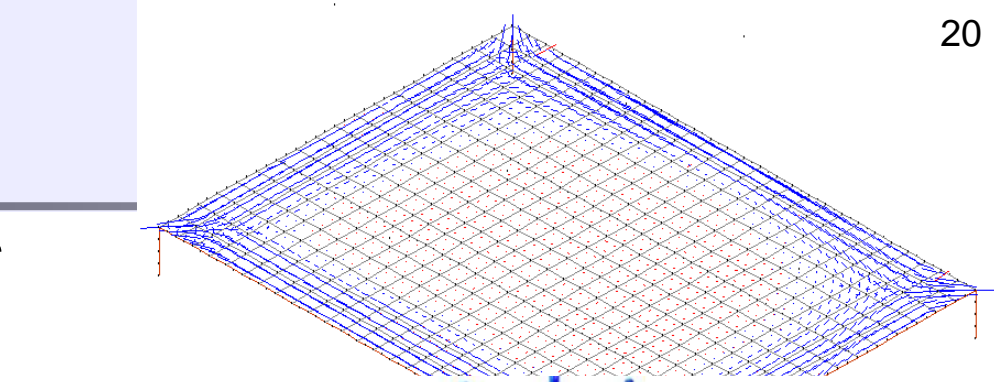
- The resistance is based on analytical relations

$$R_{fi,d,t}(X_{d,fi}) > E_{fi,d}(F_{fi,d})$$

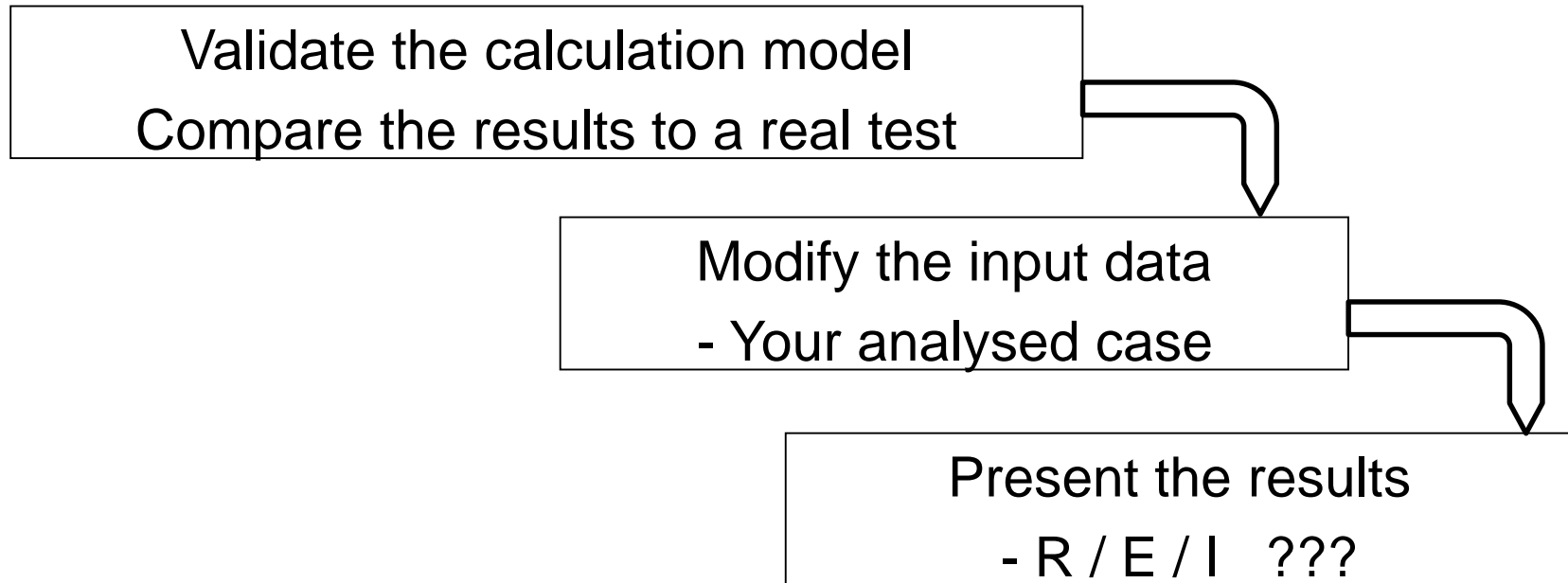
Introduction

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.

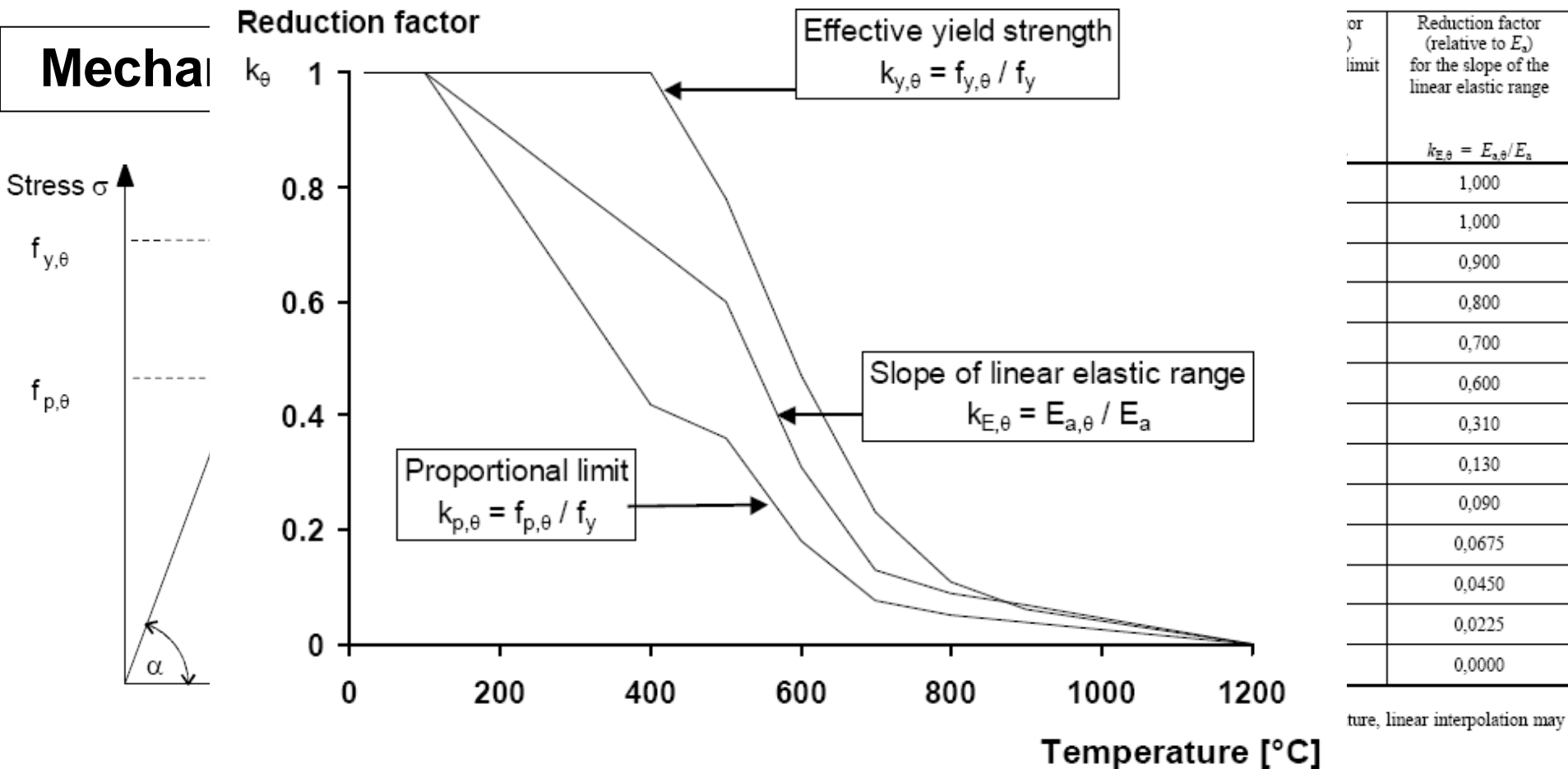


Fire design considerations



Fire design considerations

ALL CALCULATION MODELS ARE BASED ON THE MATERIAL PROPERTIES DEGRADATION CAUSED BY ELEVATED TEMPERATURE



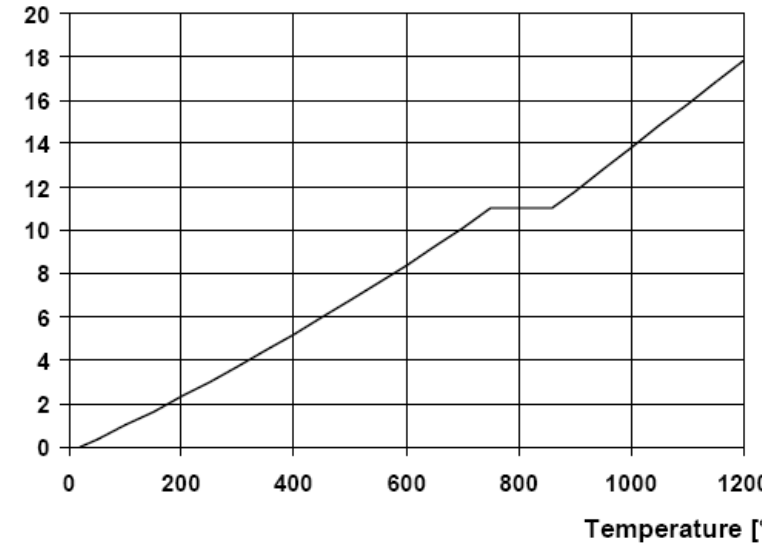
Fire design considerations

Thermal properties of steel

Thermal elongation



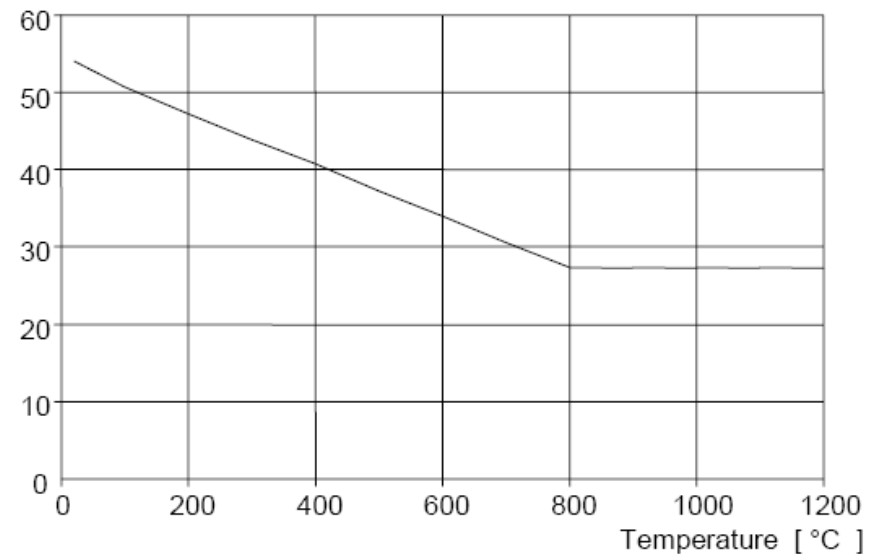
Relative Elongation $\Delta l/l$ [$\times 10^{-3}$]



Conductivity



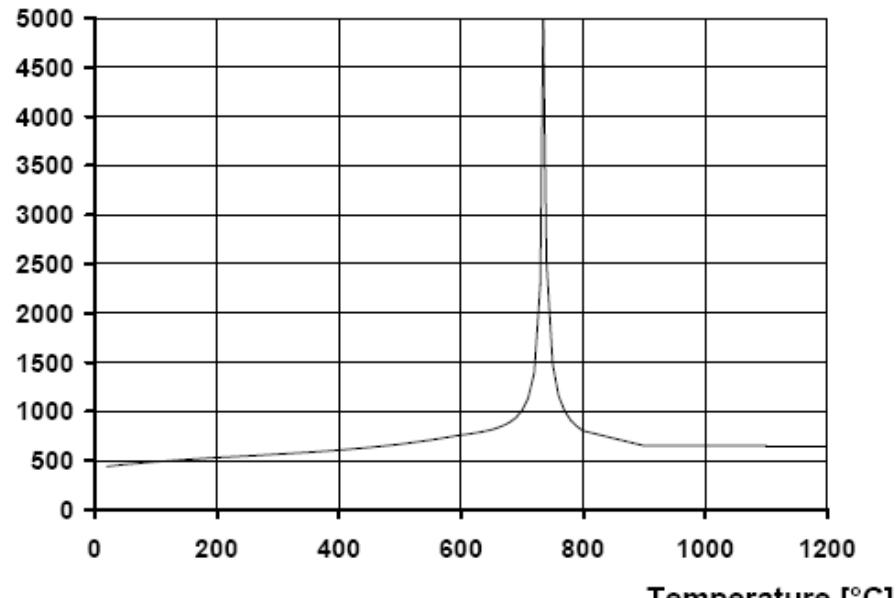
Thermal conductivity [W / mK]



Specific heat



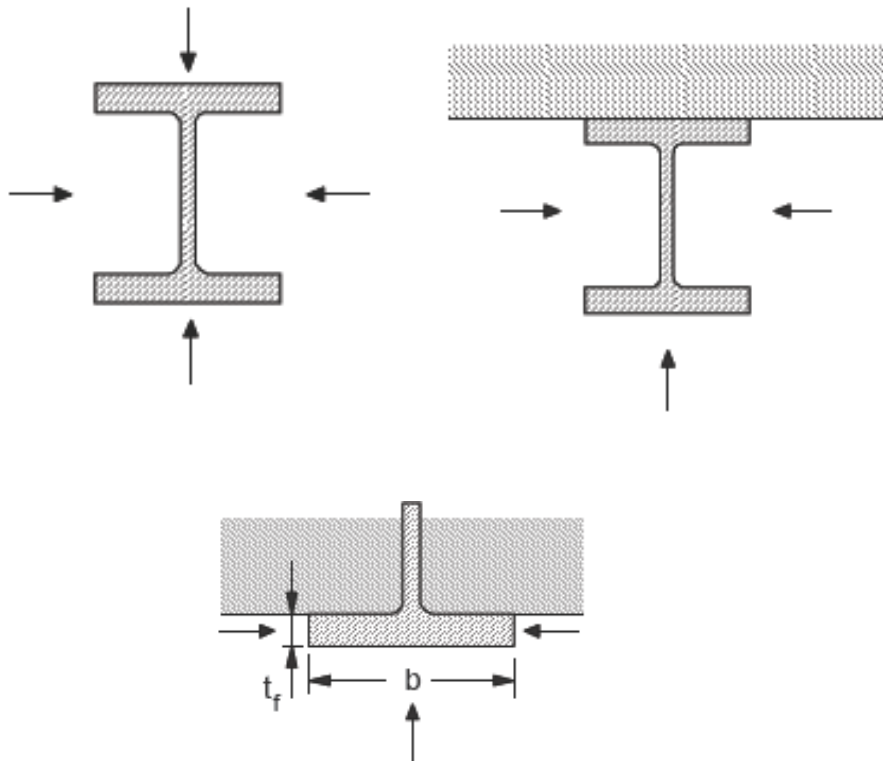
Specific heat [$\text{J} / \text{kg K}$]



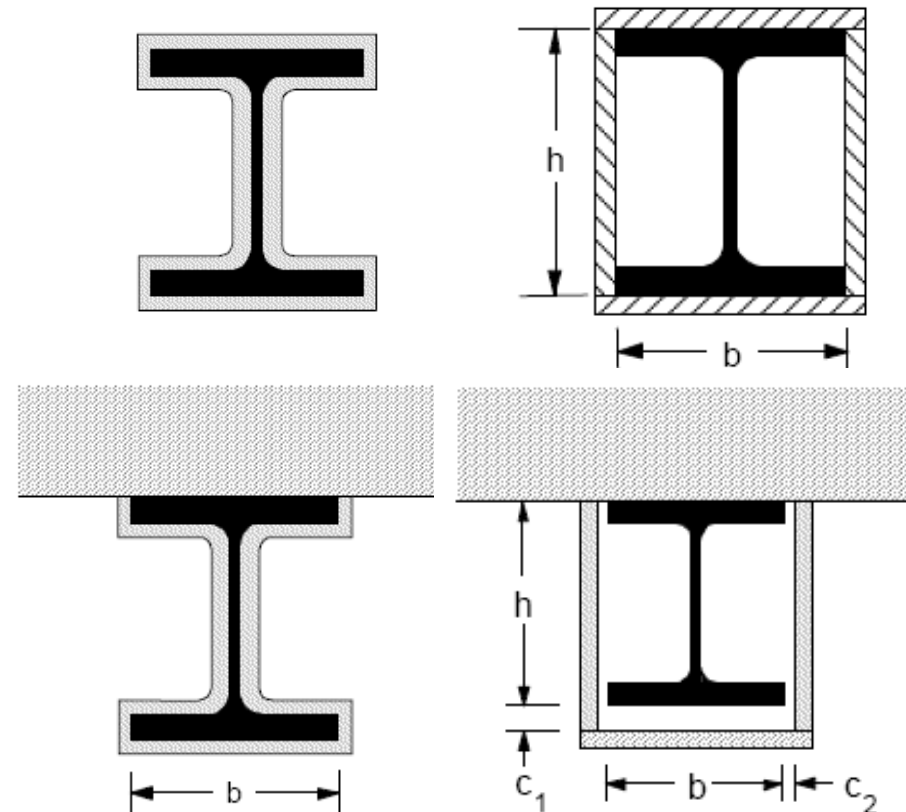
Fire design considerations

Temperature in structural elements

Unprotected steelwork



Protected steelwork

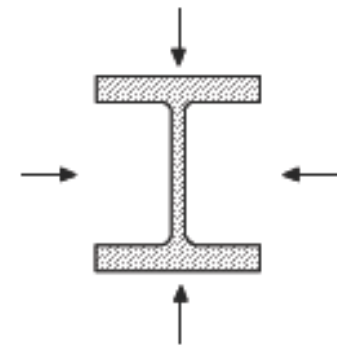


Fire design considerations

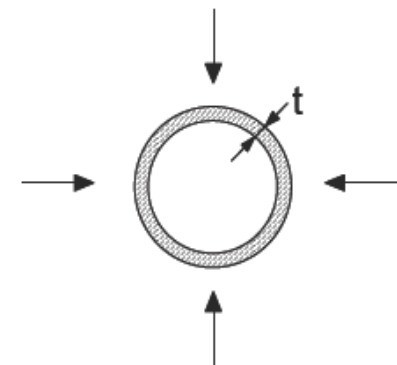
Unprotected steelwork temperature development

$$\Delta\theta_{s,t} = k_{sh} \frac{A_m/V}{c_a \rho_a} \dot{h}_{net,d} \Delta t$$

$$\frac{A_m}{V} = \frac{\text{perimeter}}{\text{cross-section area}}$$



$$A_m/V = 1/t$$



$\Delta\theta_{s,t}$ is the steel temperature increase from time t to time $t+\Delta t$,

k_{sh} is the correction factor for the shadow effect, see below,

A_m is the surface area of the member per unit length,

V is the volume of the member per unit length,

c_a is the specific heat of steel,

ρ_a is the unit mass of steel,

$\dot{h}_{net,d}$ is the design value of the net heat flux per unit area,

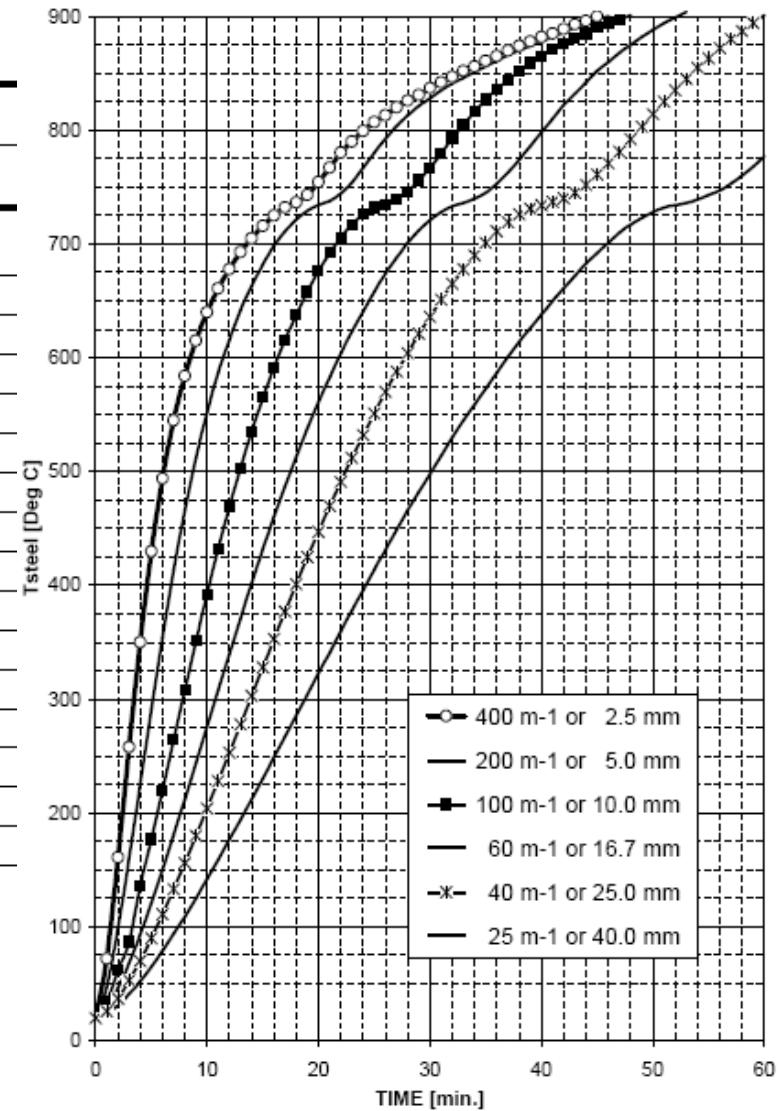
Δt is the time interval.

A_m/V : section factor

Fire design considerations

Unprotected steelwork temperature development

A_m^*/V [m^{-1}]	400	200	100	60	40	25
V/A_m^* [mm]	2.5	5.0	10.0	16.7	25.0	40.0
Time [min.]	Steel temperature in °C					
0	20	20	20	20	20	20
5	430	291	177	121	90	65
10	640	552	392	276	204	142
11	661	587	432	308	228	159
12	678	616	469	340	253	177
13	693	642	503	371	278	194
14	705	663	535	402	303	212
15	716	682	565	432	328	230
16	725	698	591	460	353	249
17	732	711	616	487	377	267
18	736	721	638	513	401	286
19	743	729	658	538	425	304
20	754	734	676	561	447	323
21	767	738	692	583	470	341
22	780	744	706	604	491	360



Fire design considerations

Protected steelwork temperature development

$$\Delta\theta_{a,t} = \frac{\lambda_p A_p / V}{d_p c_a \rho_a (1 + \phi/3)} \Delta t - (e^{\phi/10} - 1) \Delta\theta_{g,t}$$

λ_p is the thermal conductivity of the fire protection material,

A_p/V is the section factor for steel members insulated by fire protection material,

A_p is the appropriate area of fire protection material per unit length of member,

V is the volume of the member per unit length,

$\theta_{g,t}$ is the ambient gas temperature at time t ,

$\theta_{a,t}$ is the steel temperature at time t ,

d_p is the thickness of the fire protection material,

c_a is the temperature dependant specific heat of steel,

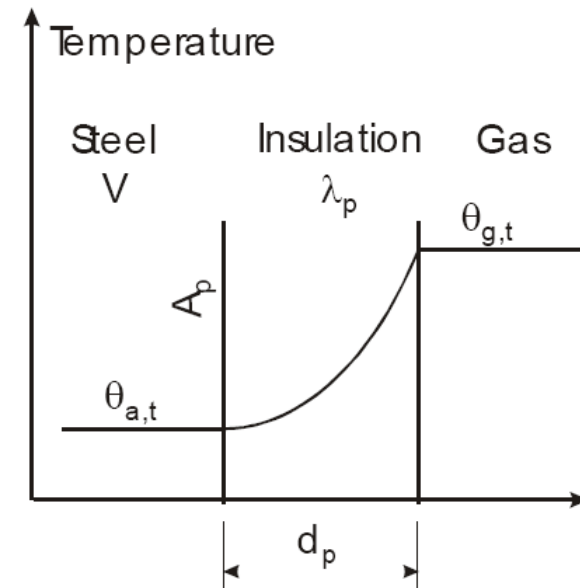
ρ_a is the unit mass of steel,

Δt is the time interval,

$\Delta\theta_{g,t}$ is the increase of the ambient gas temperature during the time interval Δt ,

c_p is the temperature independent specific heat of the fire protection material,

ρ_p is the unit mass of the fire protection material,

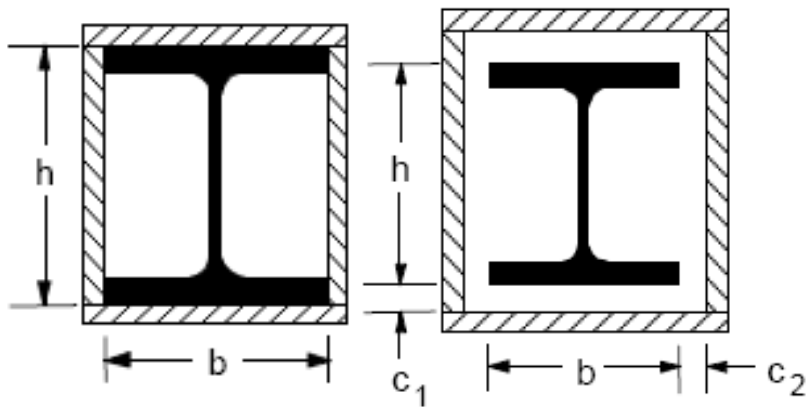


Fire design considerations

Protected steelwork temperature development

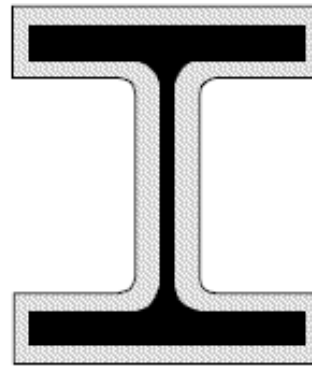
$$2(b + h)$$

steel cross-section
area



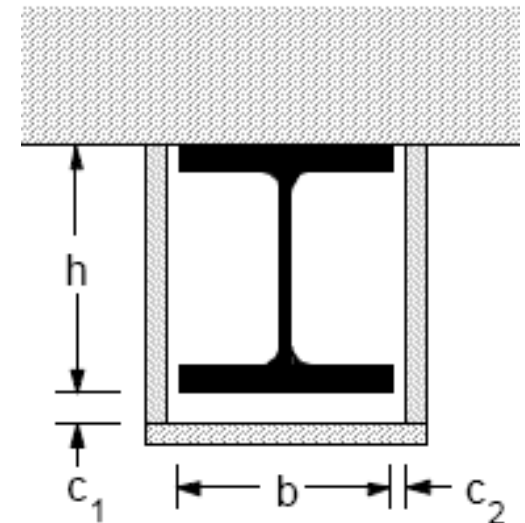
steel perimeter

steel cross-section
area



$$2h + b$$

steel cross-section
area

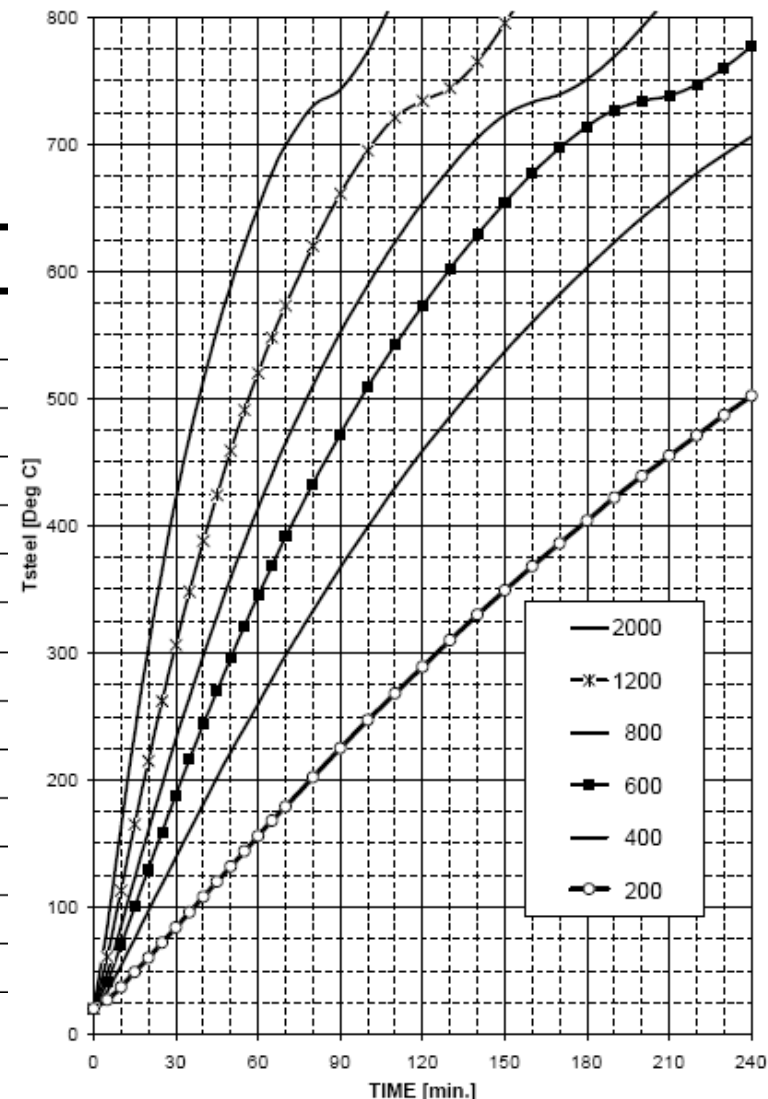


Fire design considerations

Protected steelwork temperature development

$$k_p = \frac{\lambda_p A_p}{d_p V}$$

k_p [W/m ³ K]	200	400	600	800	1200	2000
Time [min.]	Steel temperature in °C					
0	20	20	20	20	20	20
10	37	54	70	85	113	163
20	60	97	130	160	215	304
30	84	139	188	232	306	421
40	108	181	244	298	388	514
50	132	222	296	359	459	589
60	156	260	345	414	520	650
70	179	298	391	465	573	699
80	202	333	433	510	620	730
90	225	367	472	552	661	743
100	247	399	509	589	695	773
110	268	430	542	623	721	816
120	289	459	573	654	734	859



Neglecting the specific heat of insulation

Fire design considerations

Simple calculation models:

$$R_{fi,d,t}(X_{d,fi}) > E_{fi,d}(F_{fi,d})$$

- The resistance is based on analytical relations

Tensioned members

$$N_{fi,\theta,Rd} = k_{y,\theta} N_{R,d} \left[\gamma_{M,1} / \gamma_{M,fi} \right]$$

N_{Rd} - the plastic design resistance of the cross-section for normal temperature design

$k_{y,\theta}$ - the reduction factor giving the effective yield strength of steel at temperature θ reached at time t

Compressed members

$$N_{b,fi,\theta,Rd} = \chi_{fi} A k_{y,\theta} f_y / \gamma_{M,fi}$$

χ_{fi} - buckling coefficient

A - cross-section area

f_y - yield limit

$k_{y,\theta}$ - the reduction factor giving the effective yield strength of steel at temperature θ reached at time t

Fire design considerations

Simple calculatin models:

$$R_{fi,d,t}(X_{d,fi}) > E_{fi,d}(F_{fi,d})$$

- The resistance is based on analytical relations

Resistance in shear

$$V_{fi,t,Rd} = k_{y,\theta,web} V_{RD} \left[\gamma_{M,1} / \gamma_{M,fi} \right]$$

V_{Rd} - the plastic design resistance of the cross-section for normal temperature design

$k_{y\theta, web}$ - the reduction factor giving the effective yield strength of the web, at temperature θ reached at time t

Resistance in bending

$$M_{fi,\theta,Rd} = k_{y,\theta} \left[\gamma_{M,0} / \gamma_{M,fi} \right] M_{Rd}$$

χ_{fi} – buckling coefficient

W – Strength modulus

f_y – yield limit

$k_{y\theta}$ - the reduction factor giving the effective yield strength of steel at temperature θ reached at time t

$$M_{fi,\theta,Rd} = k_{y,\theta} \left[f_y / \gamma_{M,fi} \right] W$$

Summary quiz?

1 What are the basic requirements for a building to hold?

Structural materials and structural elements

2 Which are the main three types of actions?

Permanent, variable and accidental

3 What type of action is fire?

Accidental

4 How is the elevated temperature considered?

By temperature-time curves (Standard, Exterior and Hydrocarbons)

5 How is the fire resistance established from the structural point of view?

Resistance in fire > Load effect in fire

Summary quiz?

6 What are the three methods to evaluate the fire resistance in prescriptive design?

Tabulated data; Simple calculation models;
Advanced calculation models

7 What is the basic cause of structural failure in fire?

Degradation of mechanical properties

8 What is the main characteristic that influences the temperature development?

Section factor A_m/V

9 What is the principle of temperature increase for unprotected steel ?

Net heat flux (convection and radiation)

10 What is the principle of temperature increase for protected steel ?

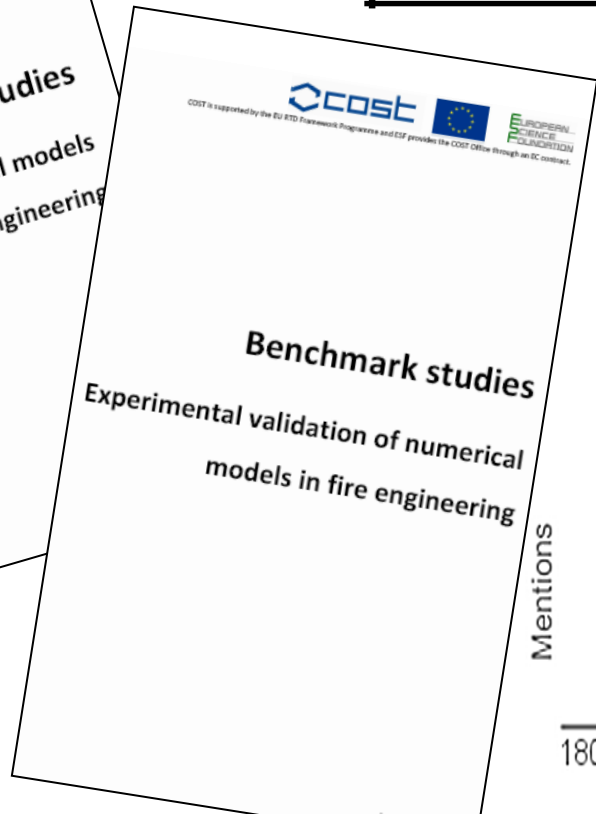
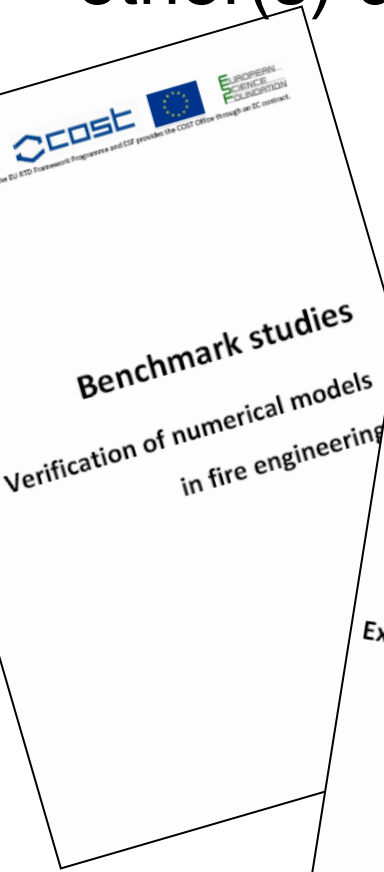
Temperature difference between insulation and steel



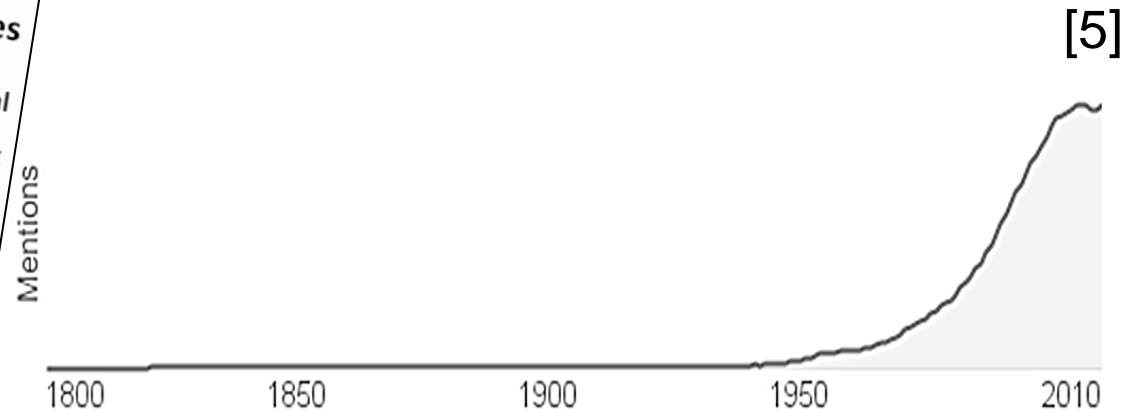
FRACOF

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



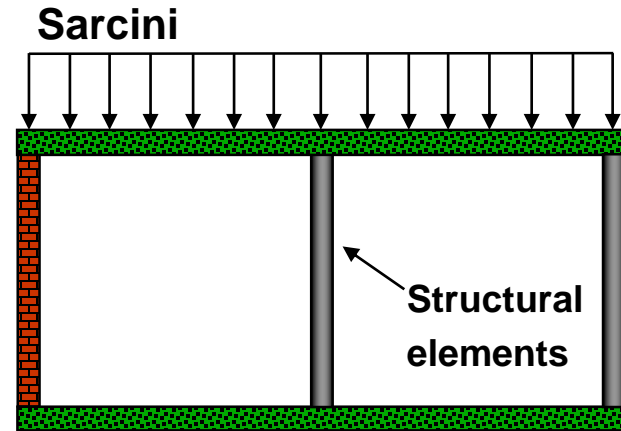
Events during fire



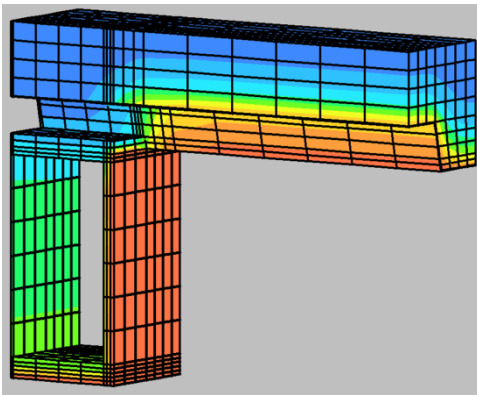
1: Ignition



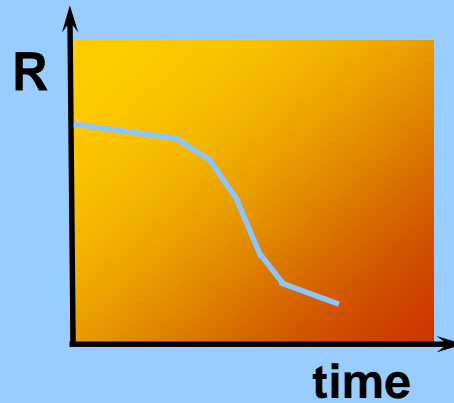
2: Thermal load



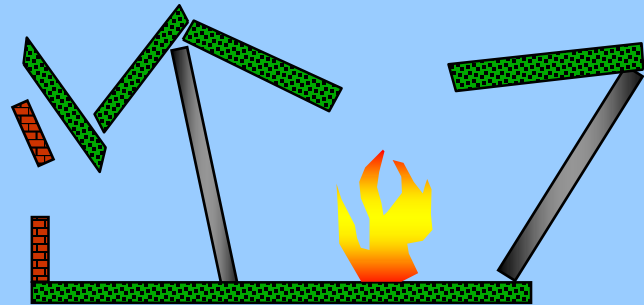
3: Mechanical load



4: Thermal response



5: Structural response



6: Failure ???

Reference case

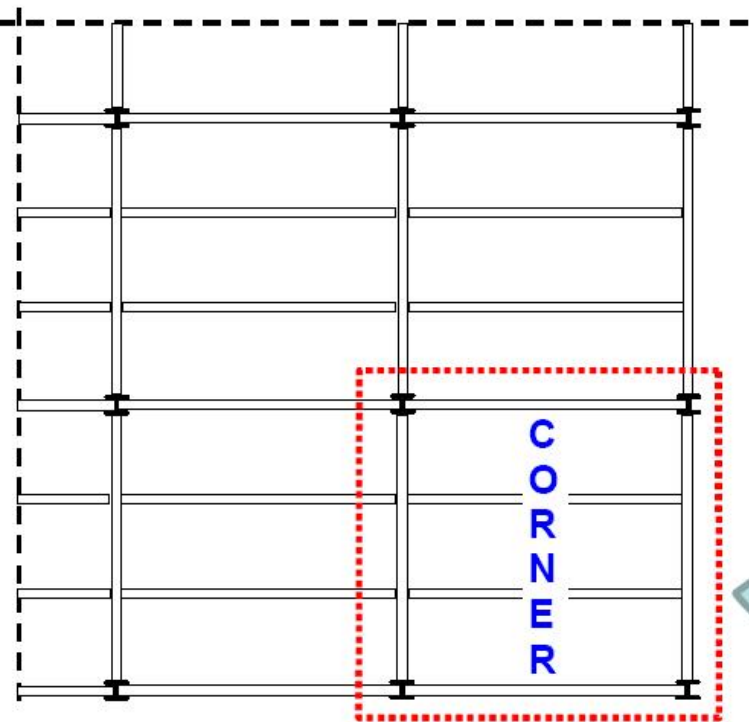
FRACOF fire test

Objectives:

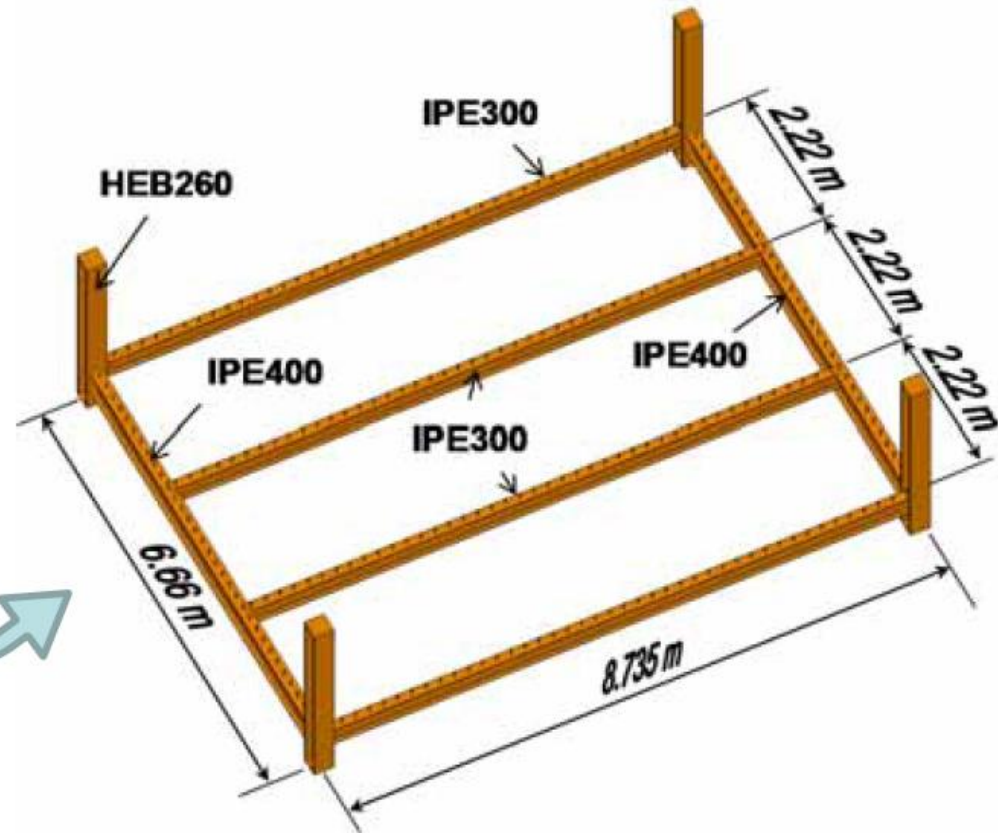
- To confirm same good performance under long fire duration (at least 90 minutes of ISO fire)
- To investigate the impact of different construction details, such as reinforcing steel mesh and fire protection of edge beams
- To validate different fire safety engineering tools

Reference case

FRACOF fire test - Setup



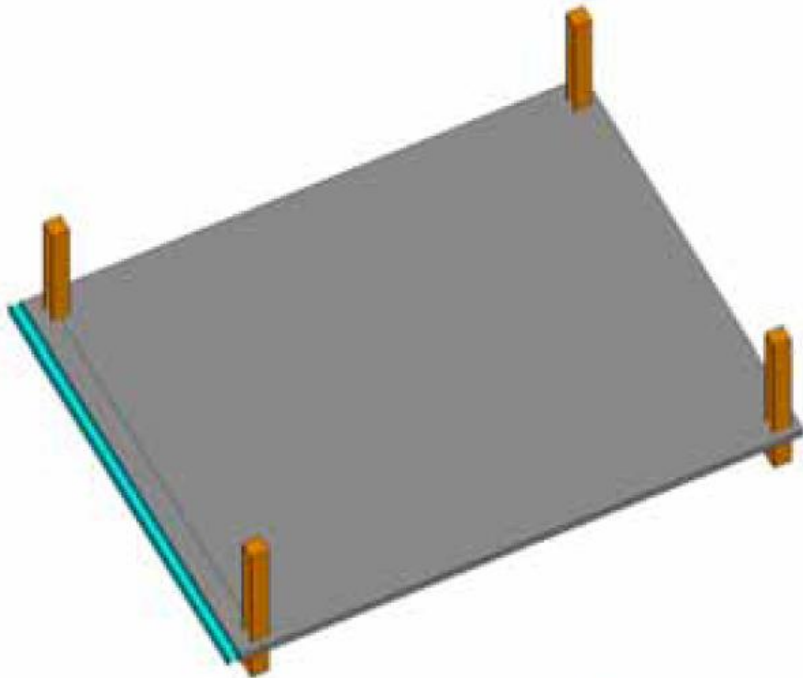
Grid of a real structure



Elements of tested structure

Reference case

FRACOF fire test - Setup



Composite floor

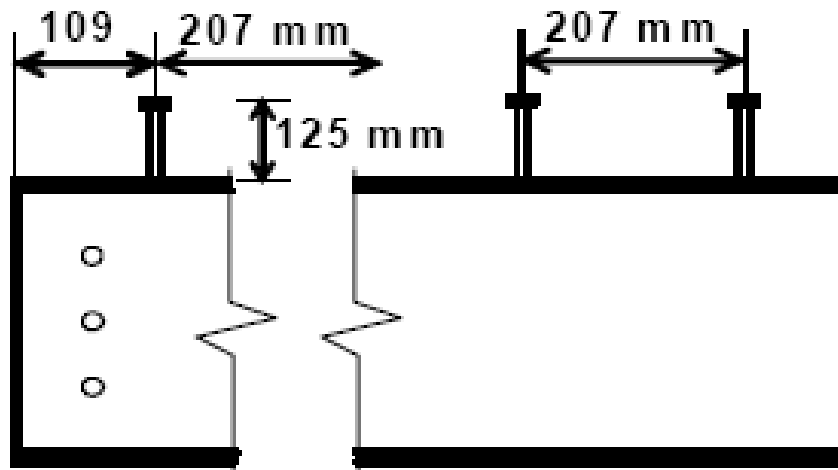


Real-scale specimen

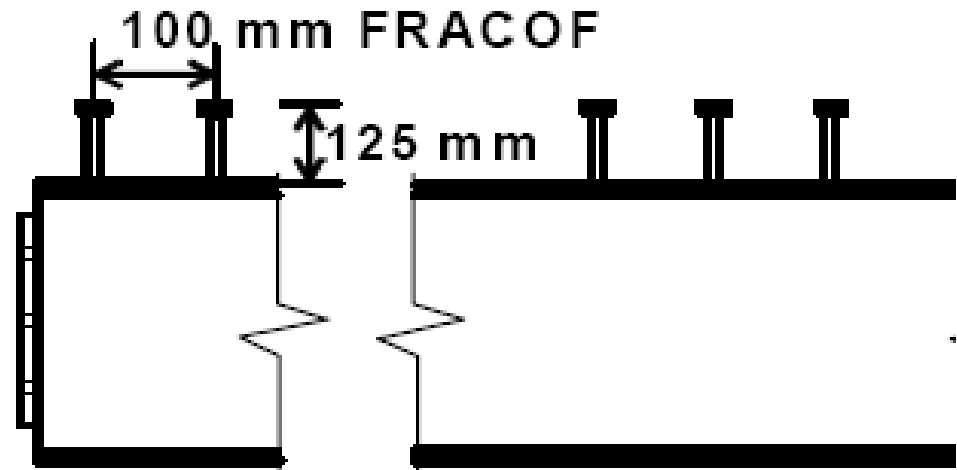
Reference case

FRACOF fire test

Beam to slab connections



Secondary beams IPE300

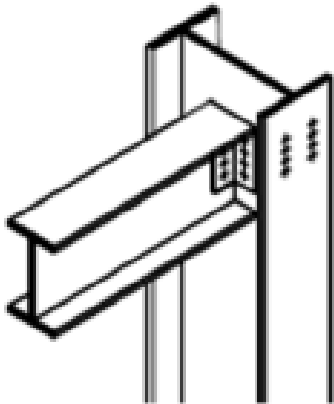
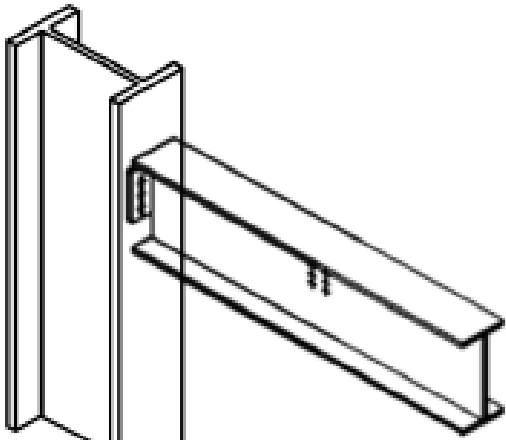
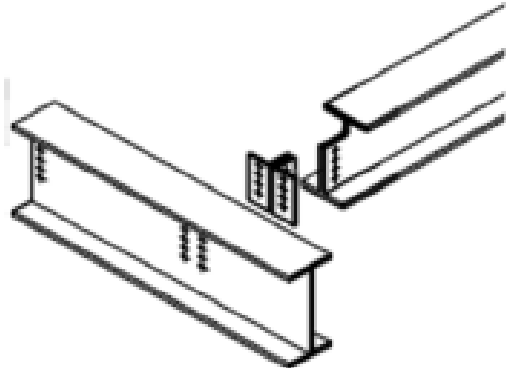


Primary beams IPE400

Full shear connection !

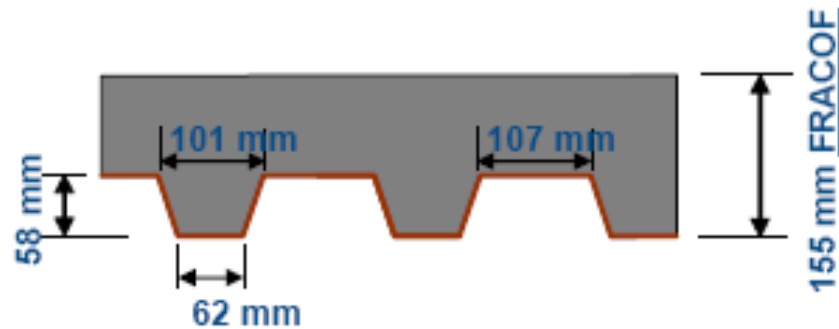
Reference case

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
		

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: $f_y=311 \text{ N/mm}^2$

Primary beams: $f_y=423 \text{ N/mm}^2$

Reinforcing steel mesh: $f_y=594 \text{ N/mm}^2$

Concrete cylinder compressive strength: $f_c=36.7 \text{ N/mm}^2$

Reference case

FRACOF fire test - Loads

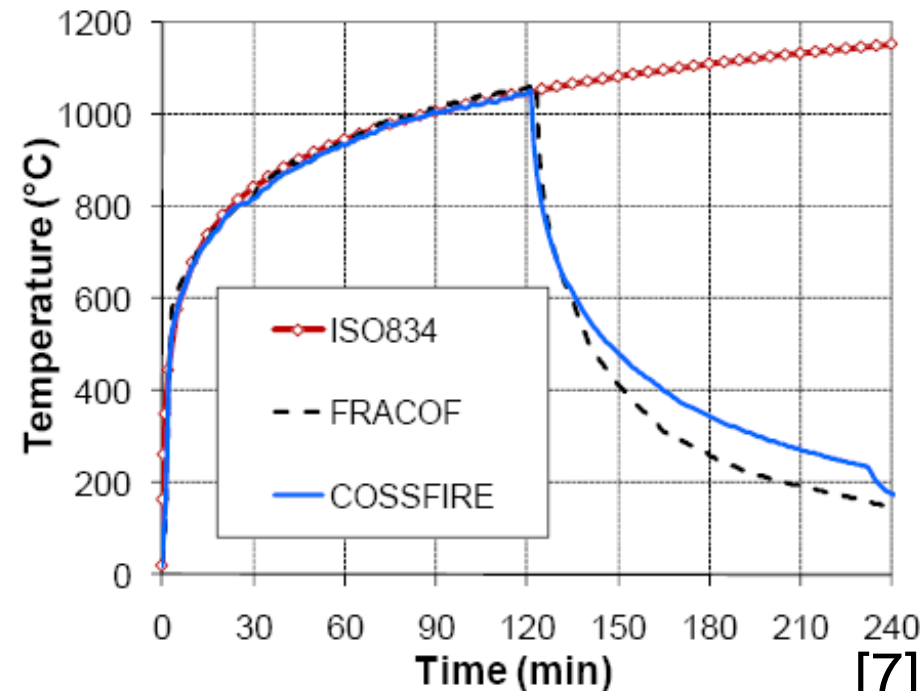
Mechanical load

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



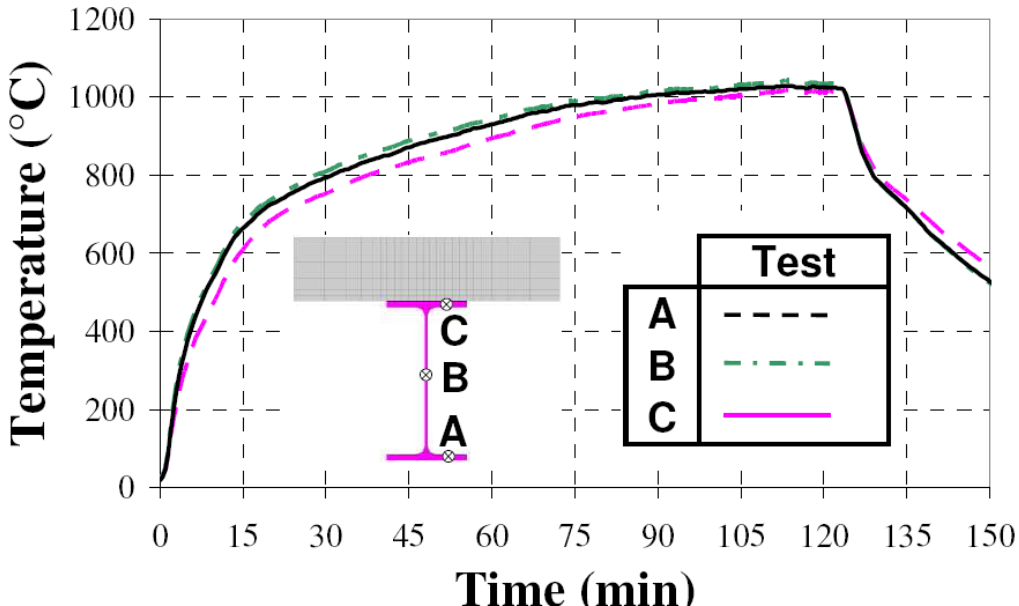
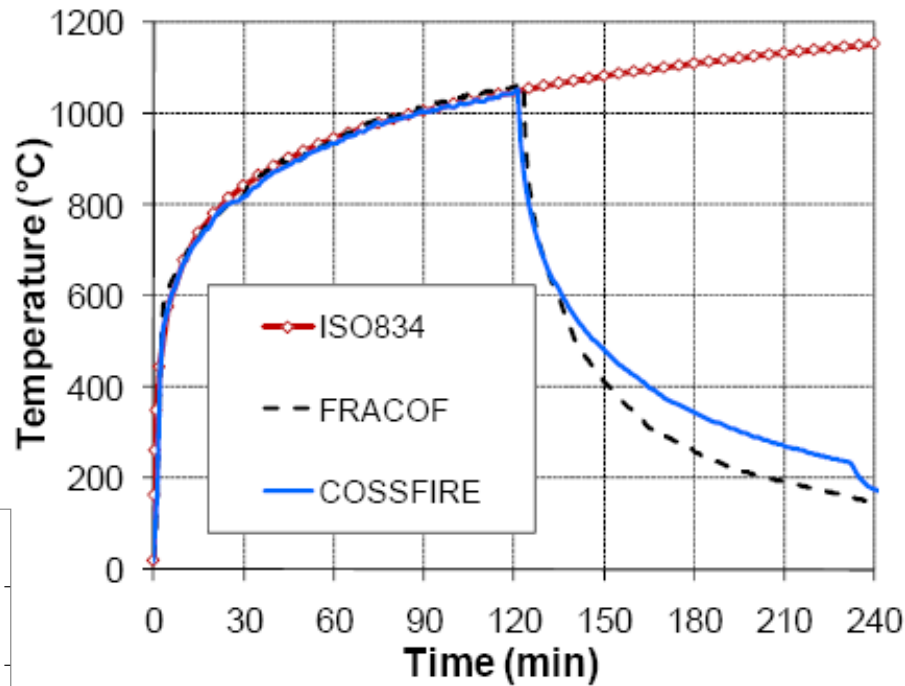
Reference case



Reference case

FRACOF fire test - Results

Fire (gas) temperature \Rightarrow

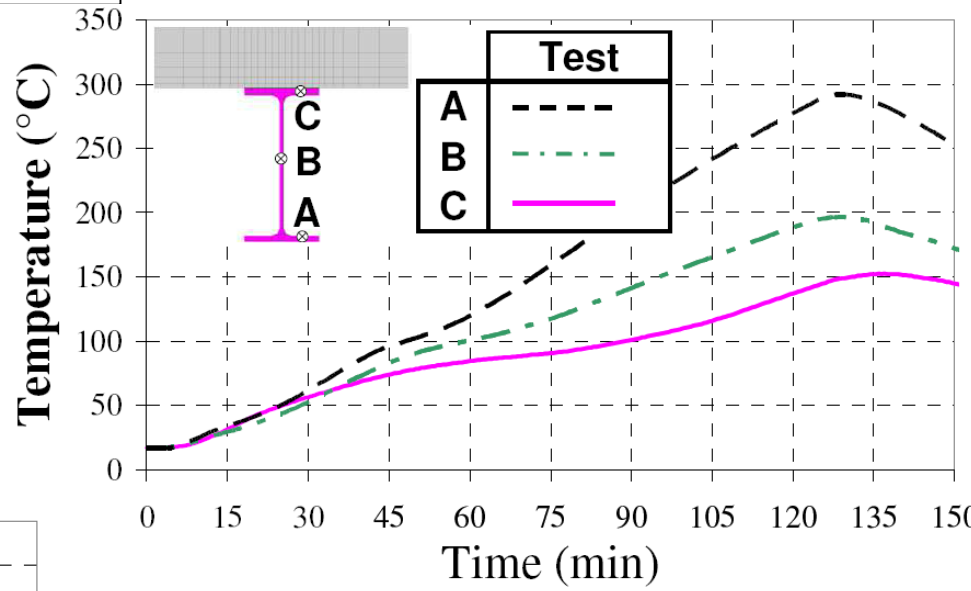


\Leftarrow Heating of unprotected steel beams

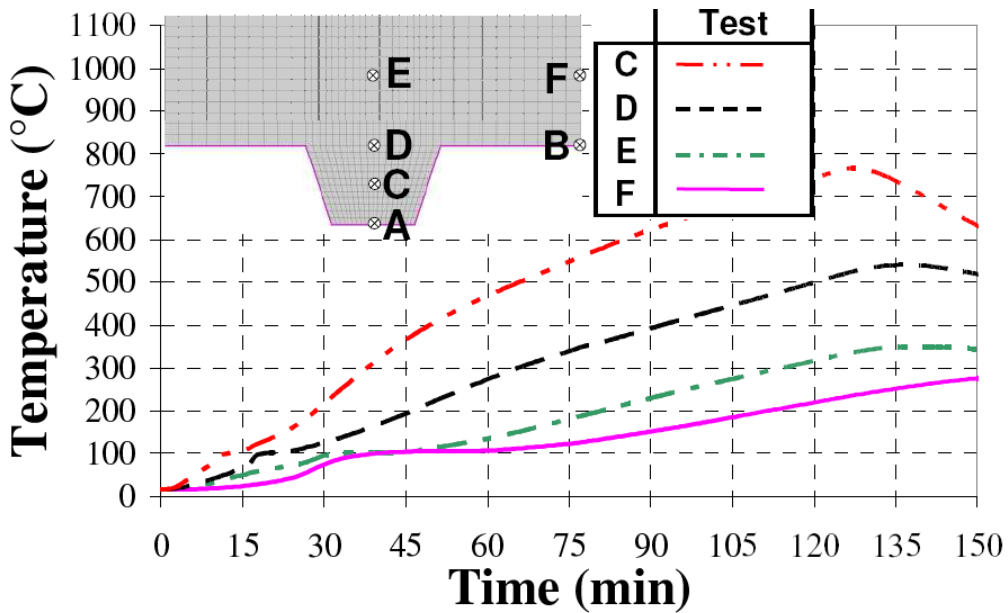
Reference case

FRACOF fire test - Results

Heating of protected steel →



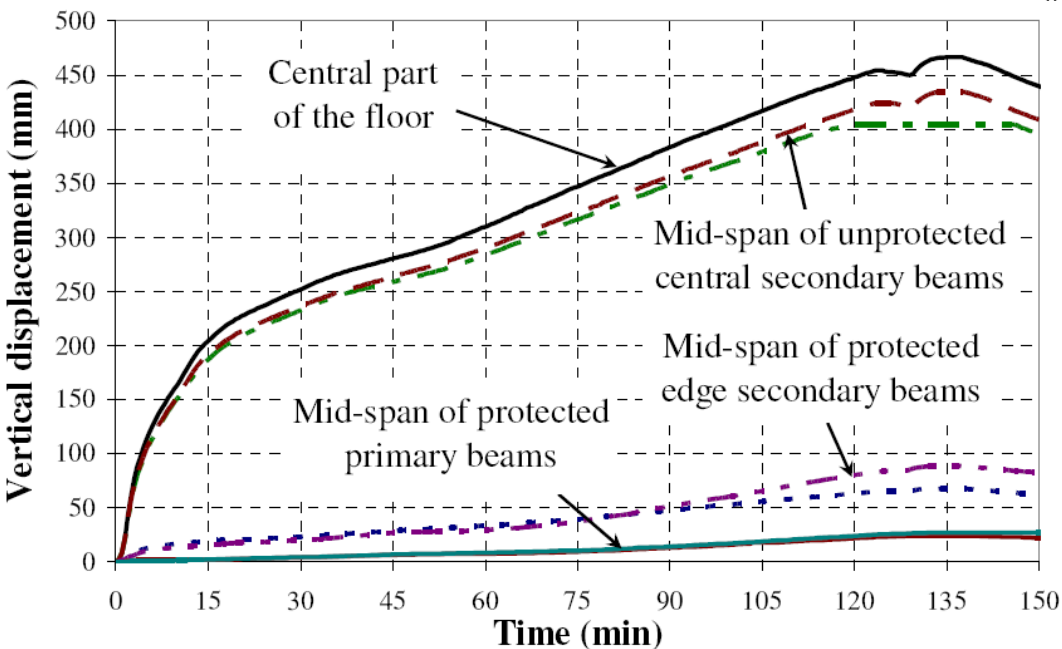
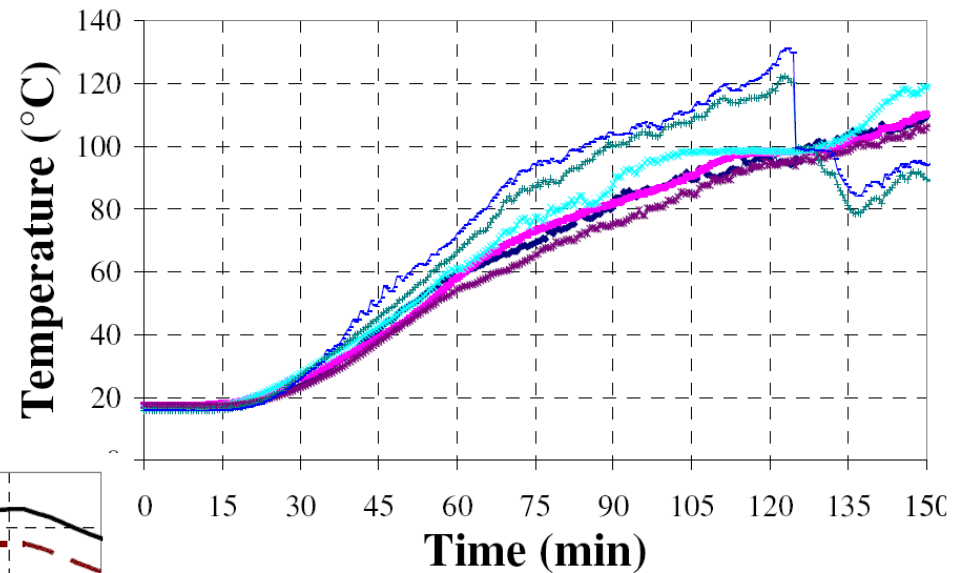
← Heating of composite slab



Reference case

FRACOF fire test - Results

Temperatures at the unexposed face of slab \Rightarrow

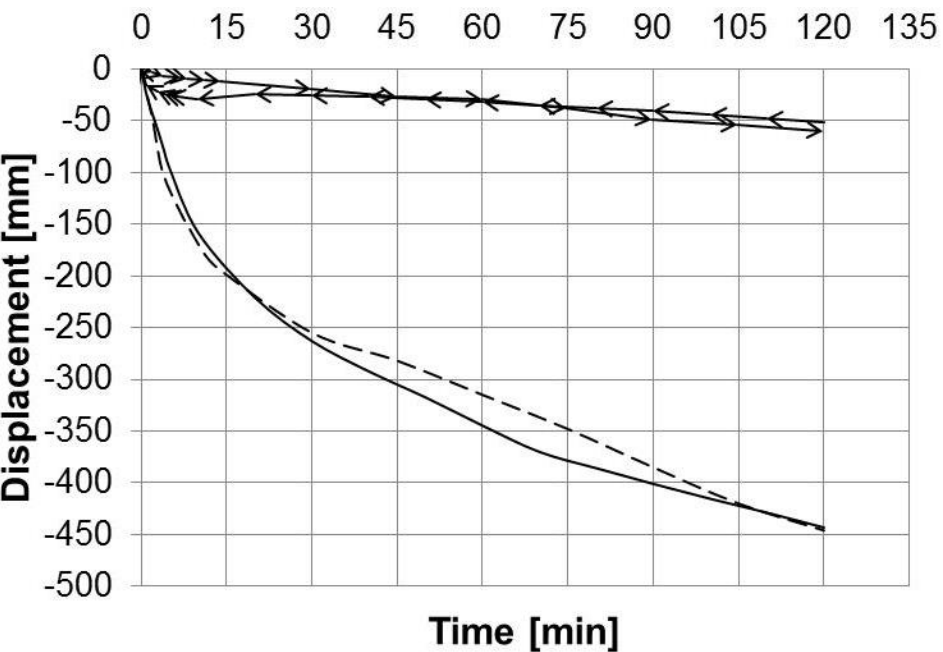


\Leftarrow Deflection of the floor

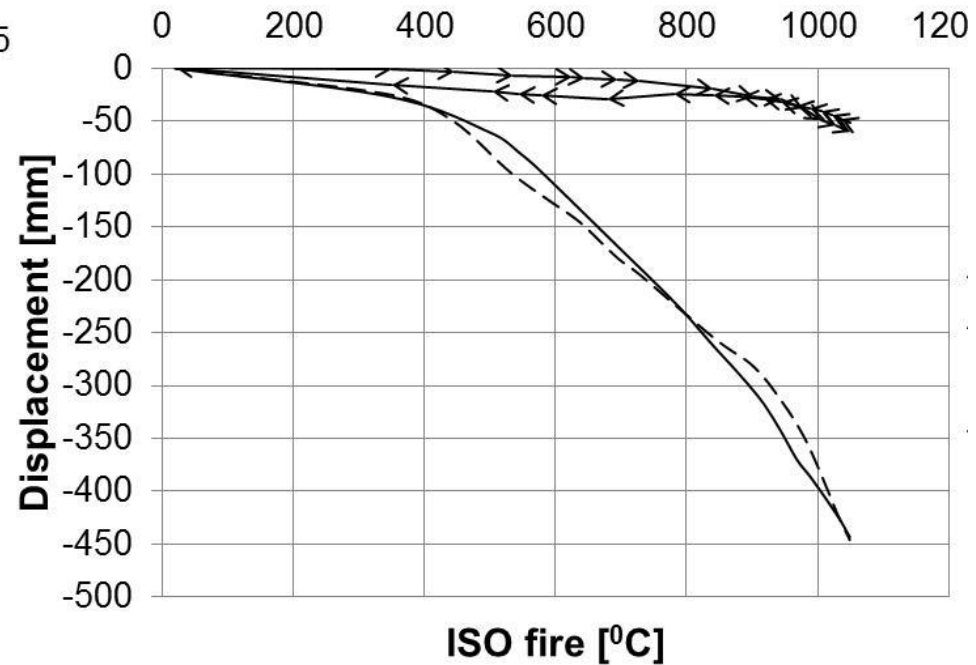
Validation

Displacement

with respect to time



with respect to temperature



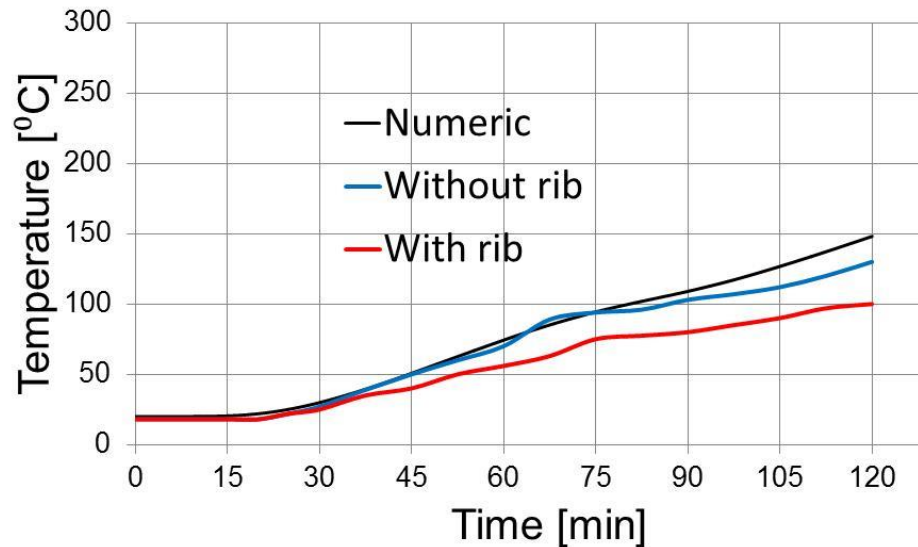
Legend:

— Central point-numeric
 -- Central Point-test

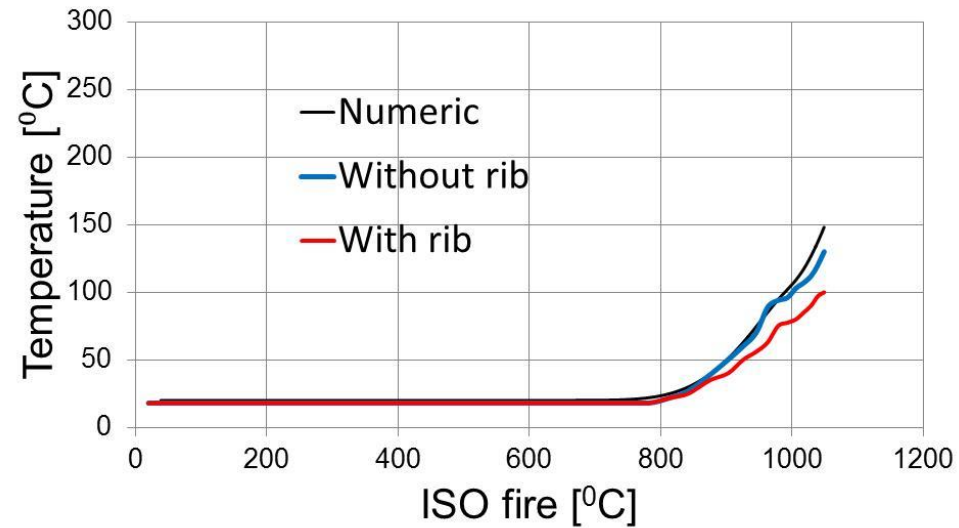
← Protected IPE300-numeric
 → Protected IPE300-test

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

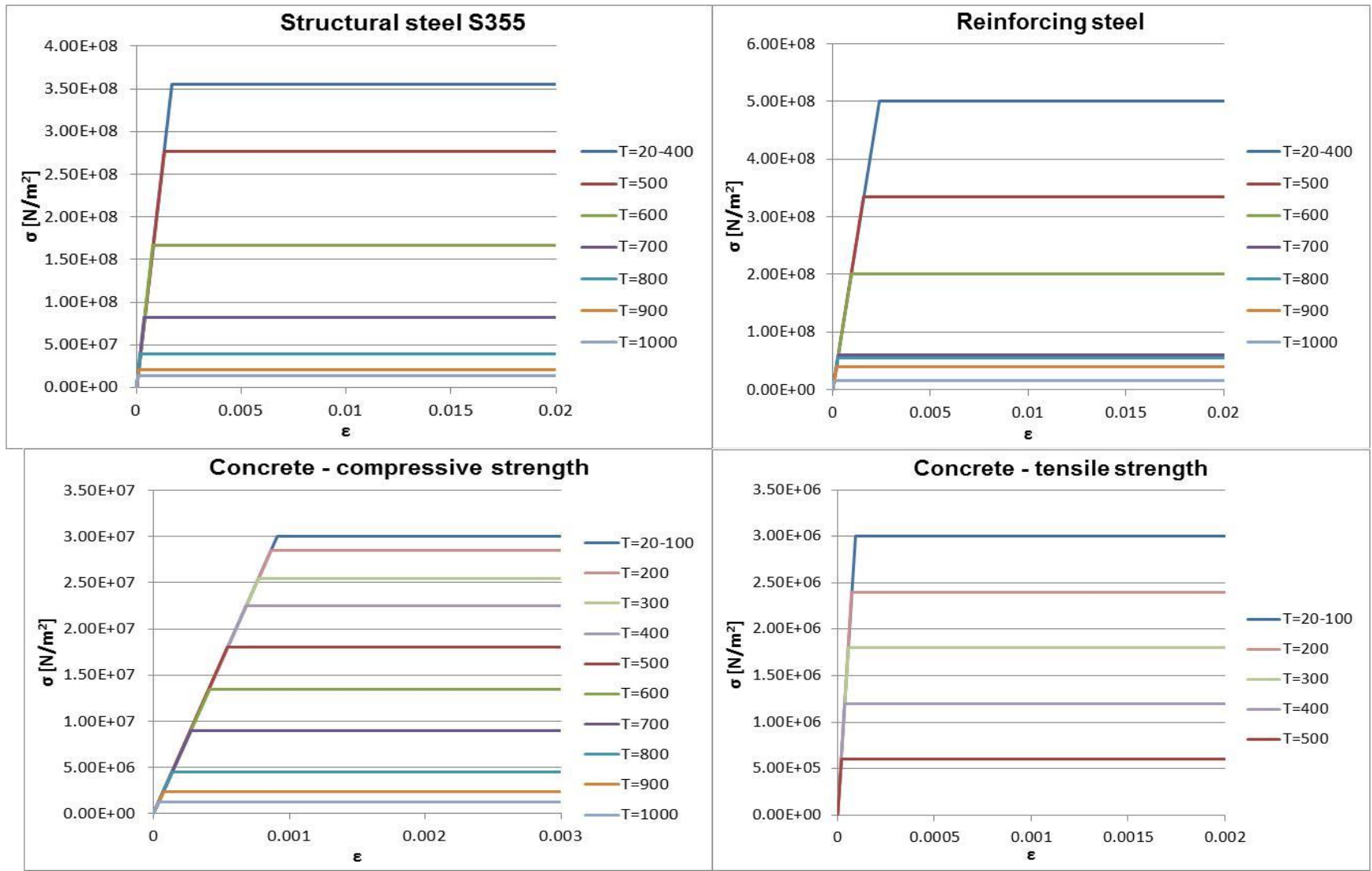
Mechanical analysis

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

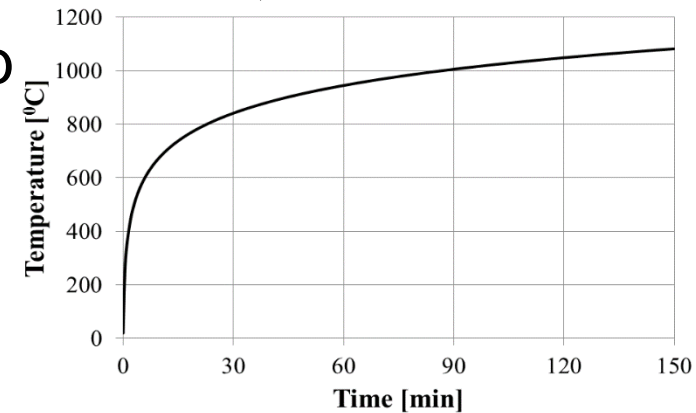


Benchmark model

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/m²
- selfweight: 3280 N/m²

7150 N/m² – uniform pressure on the slab

Heat transfer
analysis in Abaqus
summary

uncoupled heat transfer analysis

sequentially coupled thermal-stress analysis

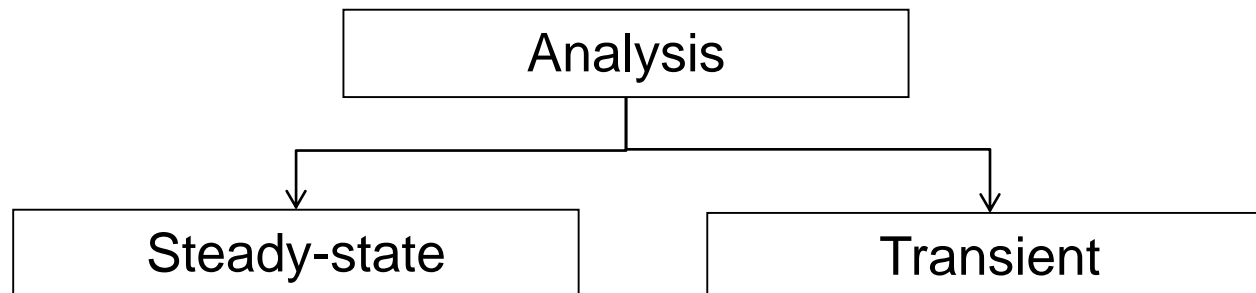
fully coupled thermal-stress analysis,

fully coupled thermal-electric-structural analysis,

adiabatic analysis,

coupled thermal-electrical analysis

cavity radiation

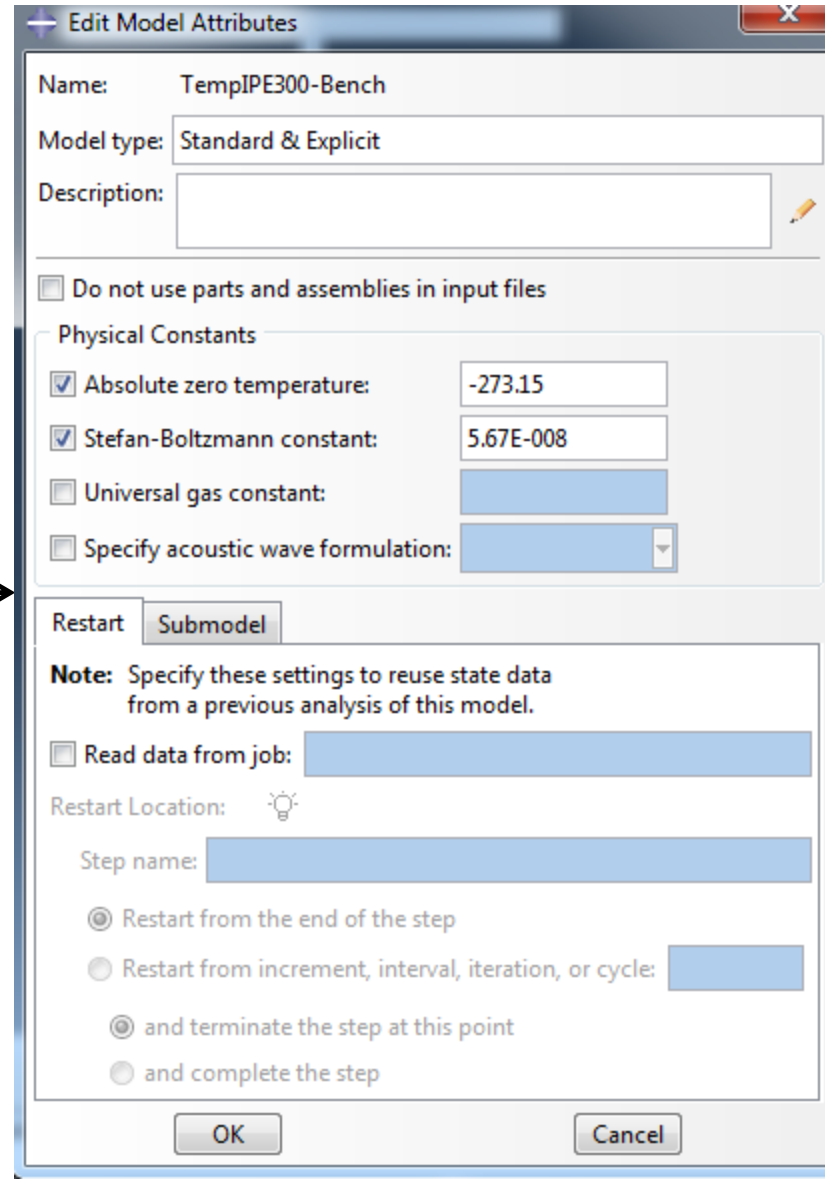


Settings

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, °C



Numerical model

Temperature field for secondary, unprotected beams

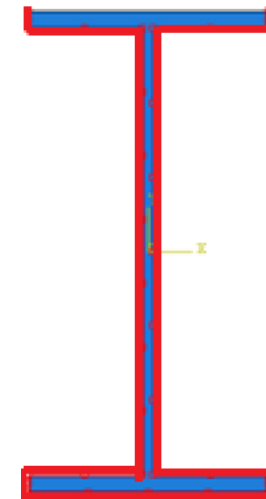
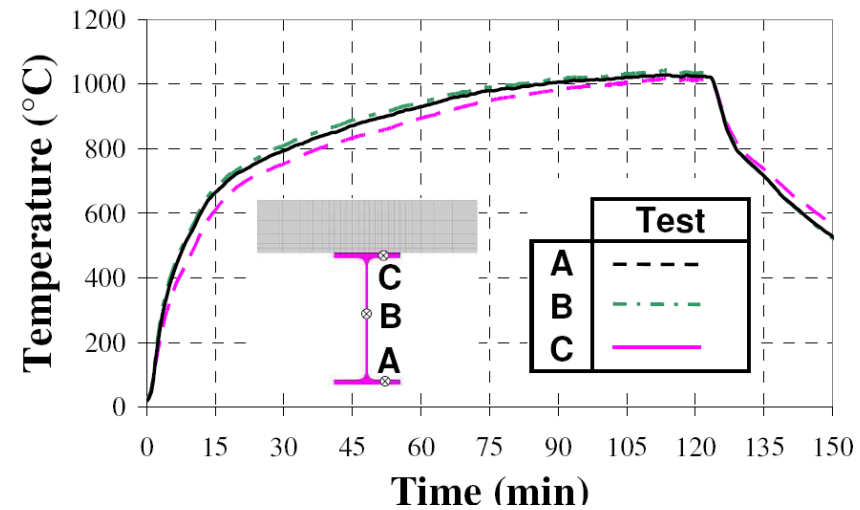
From test → uniform temperature

Create a 2D heat transfer model for the IPE300 section

Define interactions to the environment:

- convection
- radiation

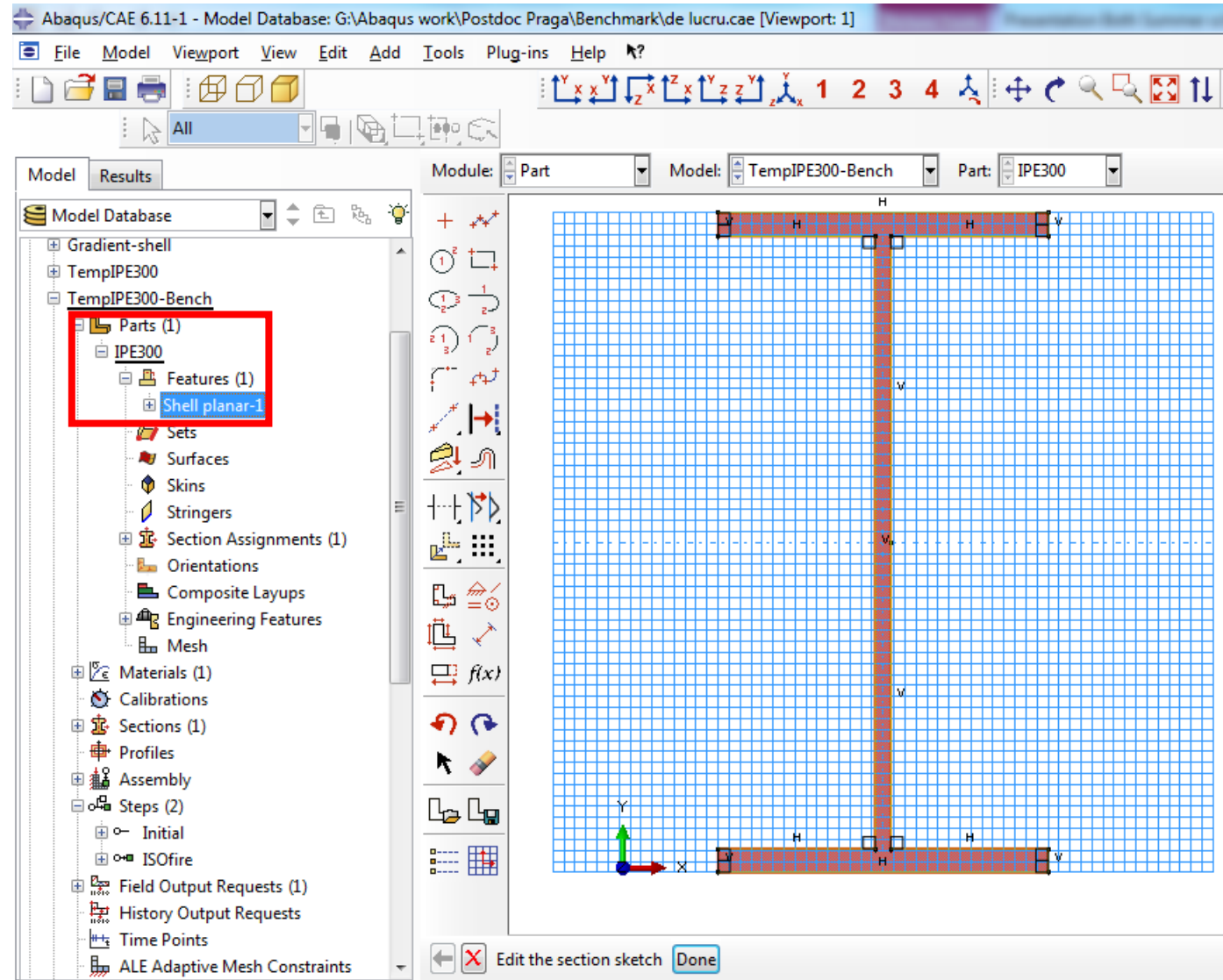
Obtain temperature field which will be used in the composite slab model



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

Abaqus/CAE 6.11-1 - Model Database: G:\Abaqus work\Postdoc Praga\Benchmark\de lucr.cae [Viewport: 1]

File Model Viewport View Material Section Profile Composite Assign Special Feature Tools Plug-ins Help

Module: Property Model: TempIPE300-Bench Part: IPE300

Model Database

- Gradient-shell
- TempIPE300
- TempIPE300-Bench
 - Parts (1)
 - Materials (1)
 - S235
 - Calibrations
 - Sections (1)
 - Profiles
 - Assembly
 - Steps (2)
 - Initial
 - ISOfire
 - Field Output Requests (1)
 - History Output Requests
 - Time Points
 - ALE Adaptive Mesh Constraints
 - Interactions (2)
 - Interaction Properties
 - Contact Controls
 - Contact Initializations
 - Constraints
 - Connector Sections
 - Fields
 - Amplitudes (3)
 - Loads
 - BCs
 - Predefined Fields (1)

Edit Material

Name: S235

Description:

Material Behaviors

- Conductivity
- Density
- Specific Heat

General Mechanical Thermal Other

Specific Heat

Type: Constant Volume Constant Pressure

Use temperature-dependent data

Number of field variables: 0

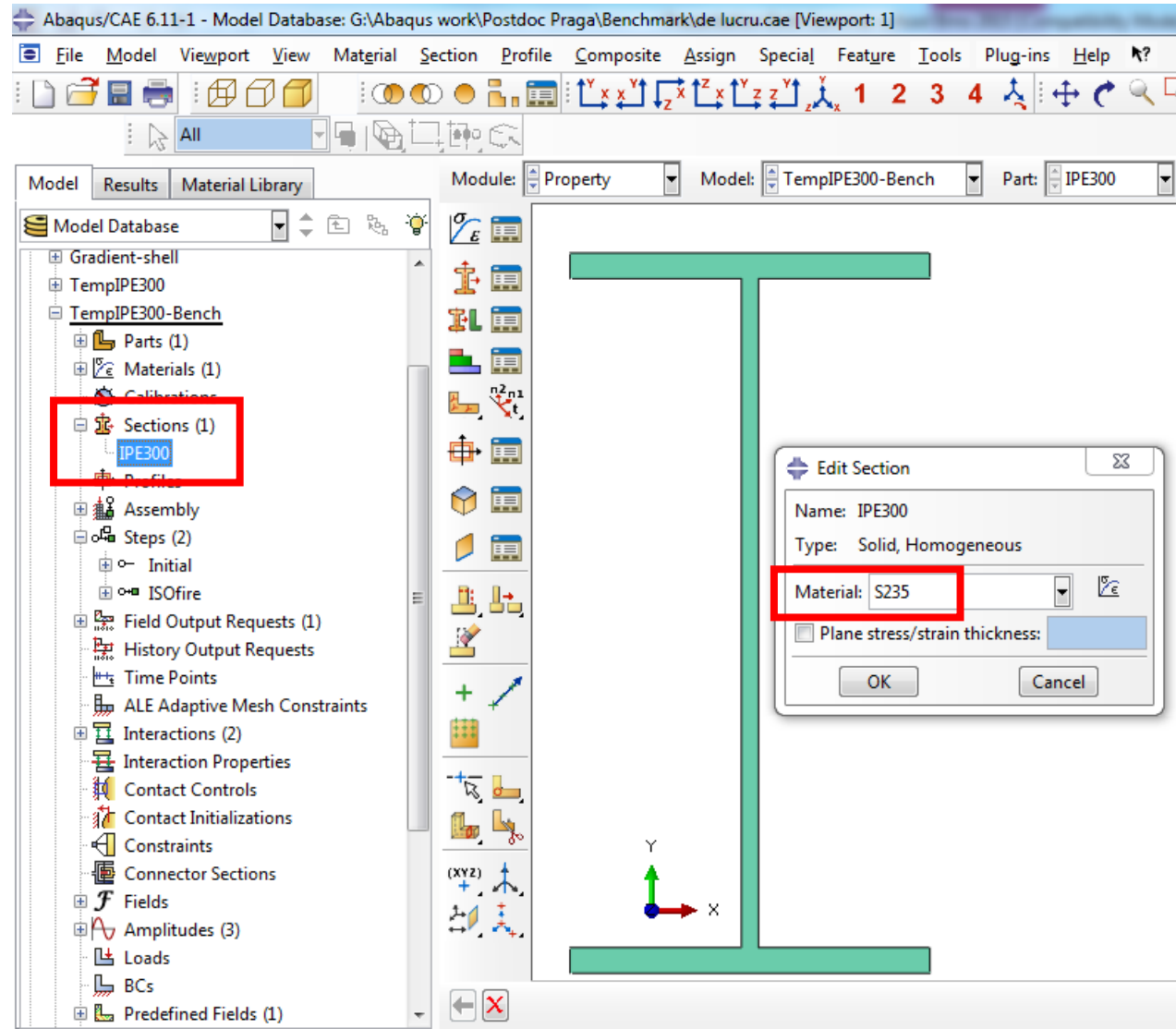
Data

	Specific Heat
1	550

Numerical model

Temperature field for secondary, unprotected beams

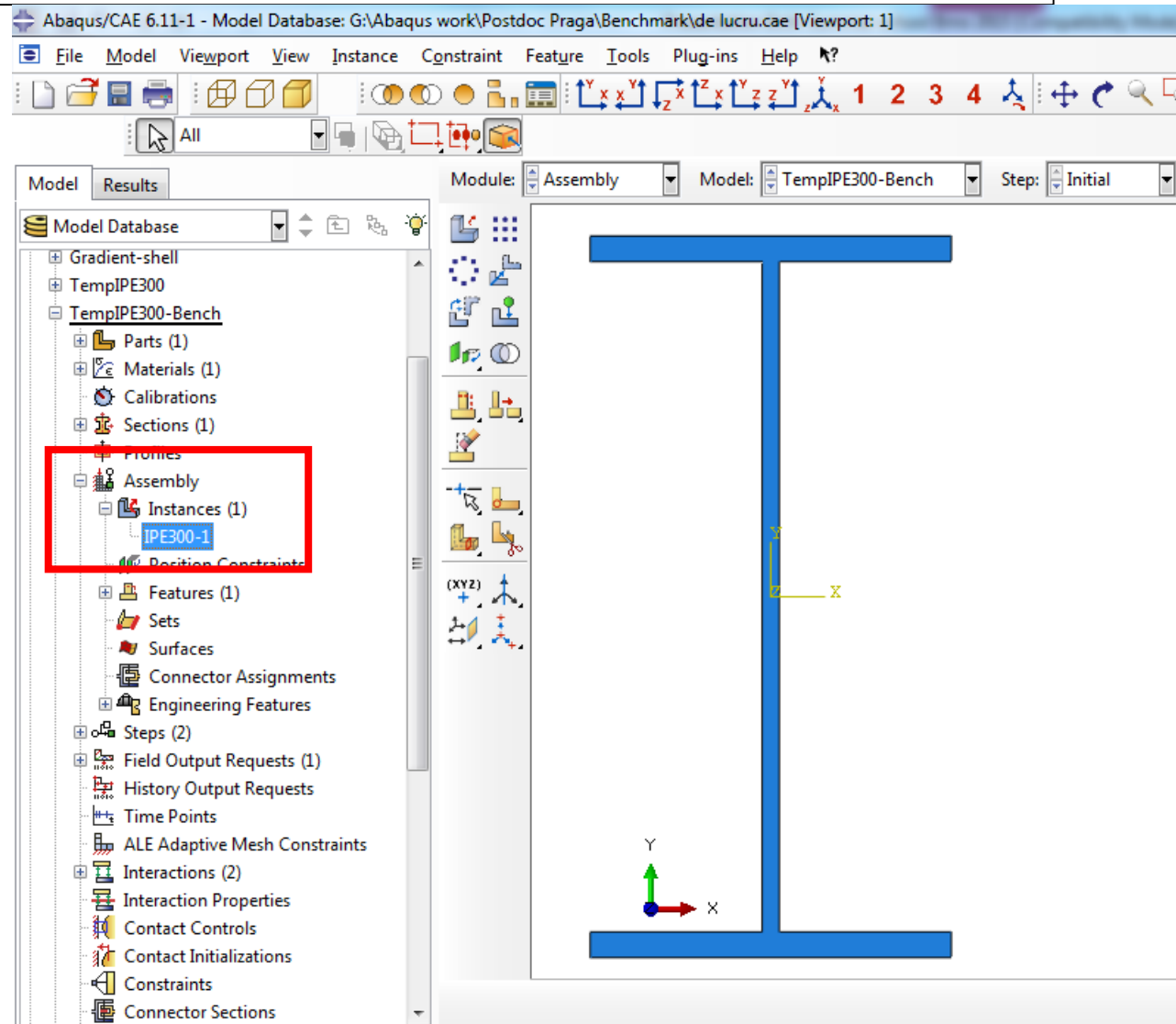
Define section property:
- Solid homogeneous



Numerical model

Temperature field for secondary, unprotected beams

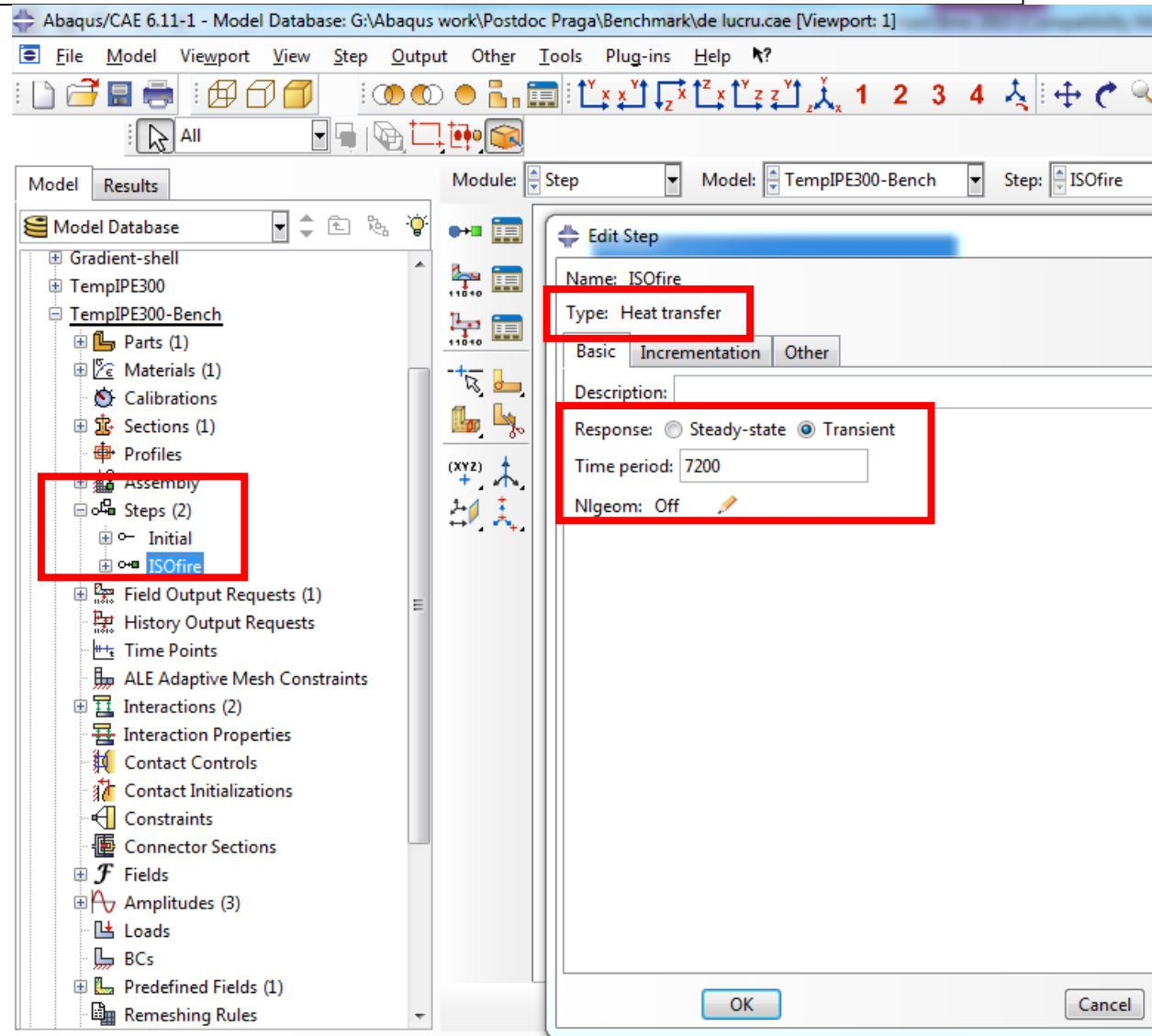
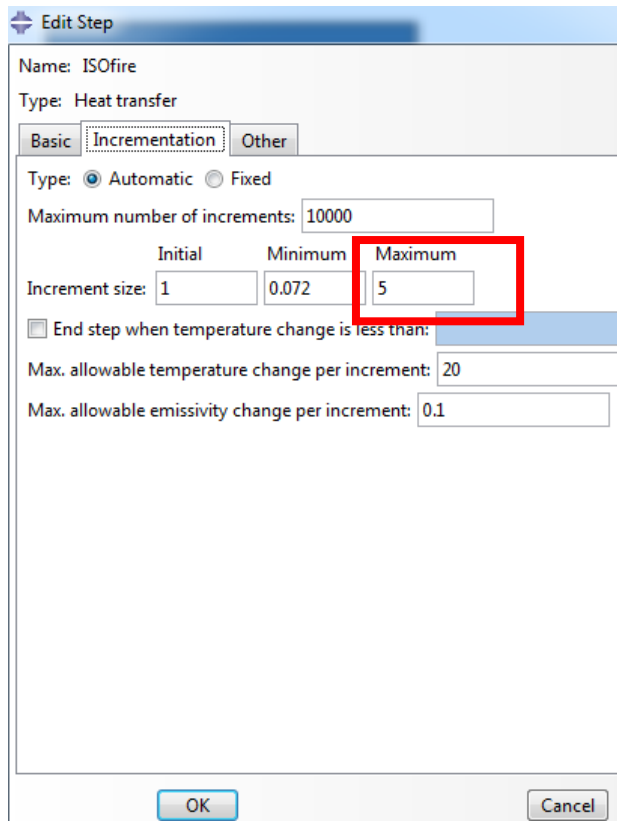
Create instances:
- IPE300



Numerical model

Temperature field for secondary, unprotected beams

Define steps:
- Heat transfer



Numerical model

Temperature field for secondary, unprotected beams

Define fire curve:
- Amplitude

Abaqus/CAE 6.11-1 - Model Database: G:\Abaqus work\Postdoc Praga\Benchmark\de lucru.cae [Viewport: 1]

File Model Viewport View Interaction Constraint Connector Special Feature Tools Plug-ins Help

Module: Interaction Model: TempIPE300-Bench Step:

Model Database

- Parts (1)
- Materials (1)
- Calibrations
- Sections (1)
- Profiles
- Assembly
- Steps (2)
- Field Output Requests (1)
- History Output Requests
- Time Points
- ALE Adaptive Mesh Constraints
- Interactions (2)
- Interaction Properties
- Contact Controls
- Contact Initializations
- Constraints
- Connector Sections
- Fields
 - Amplitudes (3)
 - FOC-ISO-C**
 - LINEAR ION
- MAINBEAMTEMP
- Loads
- BCs
- Predefined Fields (1)
- Remeshing Rules
- Optimization Tasks
- Sketches

Edit Amplitude

Name: FOC-ISO-C

Type: **Tabular**

Time span: Step time

Smoothing: Use solver default
 Specify: []

Amplitude Data

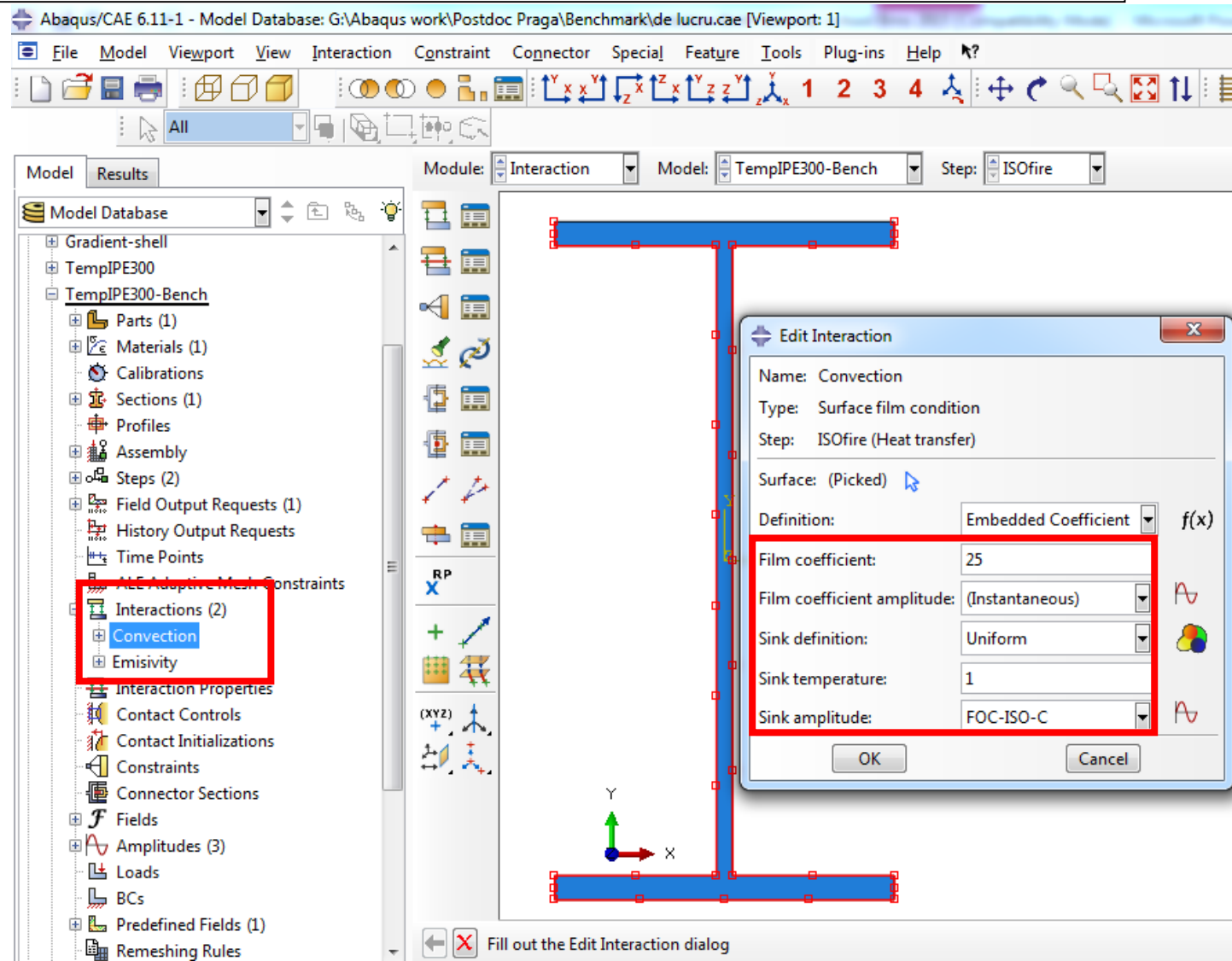
	Time/Frequency	Amplitude
1	0	20
2	30	261.14
3	60	349.21
4	120	444.5
5	180	502.29
6	240	543.89
7	300	576.41
8	360	603.12
9	420	625.78
10	480	645.46
11	540	662.85
12	600	678.43
13	660	692.54

OK Cancel

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)

The screenshot shows the Abaqus/CAE 6.11-1 interface. The main window displays a 3D model of a T-shaped beam structure. The 'Edit Interaction' dialog box is open, showing the following settings:

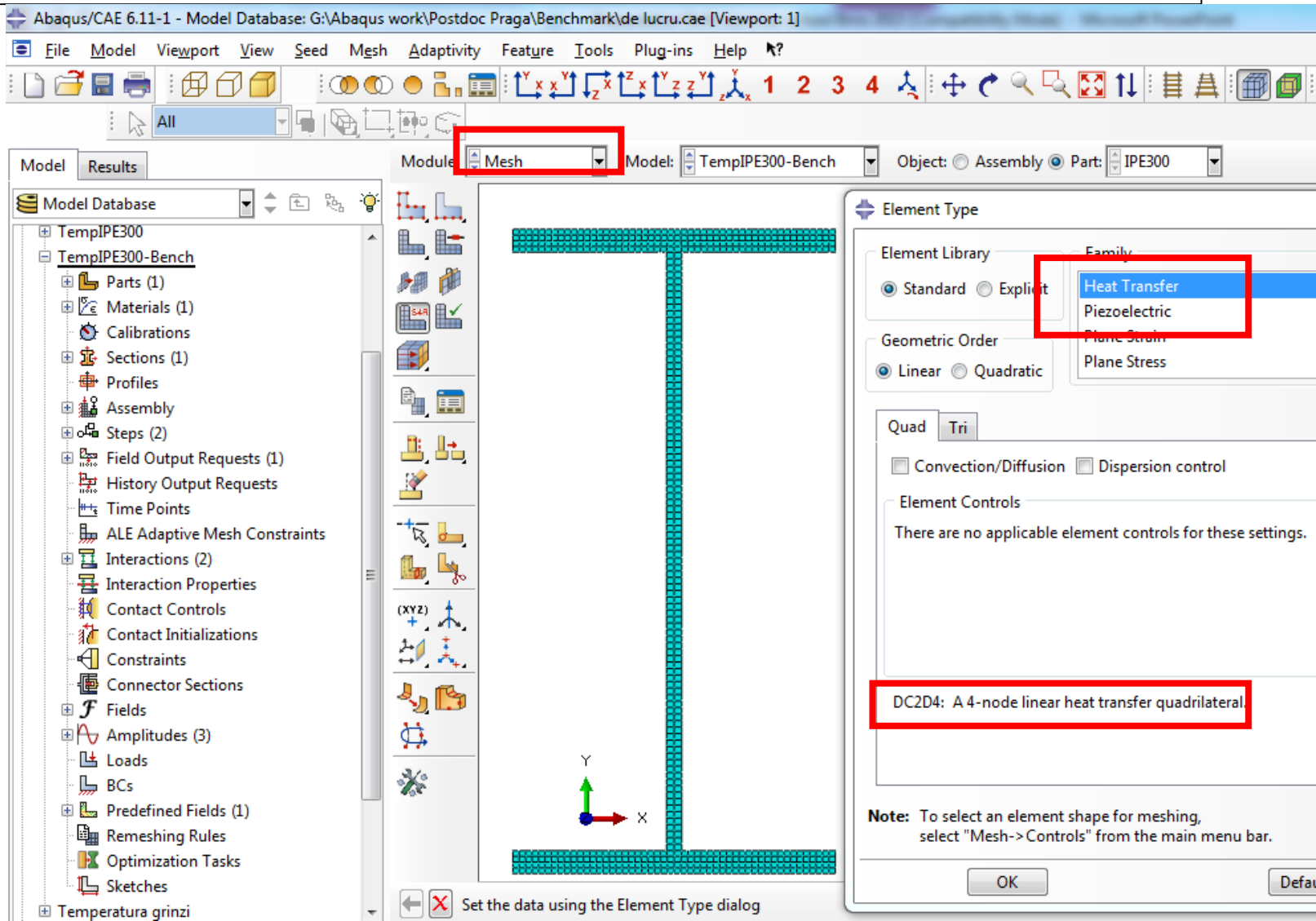
- Name: Emissivity
- Type: Surface radiation
- Step: ISOfire (Heat transfer)
- Surface: (Picked)
- Radiation type: To ambient Cavity approximation (3D only)
- Emissivity distribution: Uniform $f(x)$
- Emissivity: 0.7
- Ambient temperature: 1
- Ambient temperature amplitude: FOC-ISO-C

The 'Edit Interaction' dialog box is highlighted with a red border. The 'Emissivity distribution' dropdown is set to 'Uniform' with a small $f(x)$ icon to its right. The 'Emissivity' field contains the value '0.7'. The 'Ambient temperature' field contains the value '1'. The 'Ambient temperature amplitude' dropdown is set to 'FOC-ISO-C'. The 'OK' and 'Cancel' buttons are visible at the bottom of the dialog box.

Numerical model

Temperature field for secondary, unprotected beams

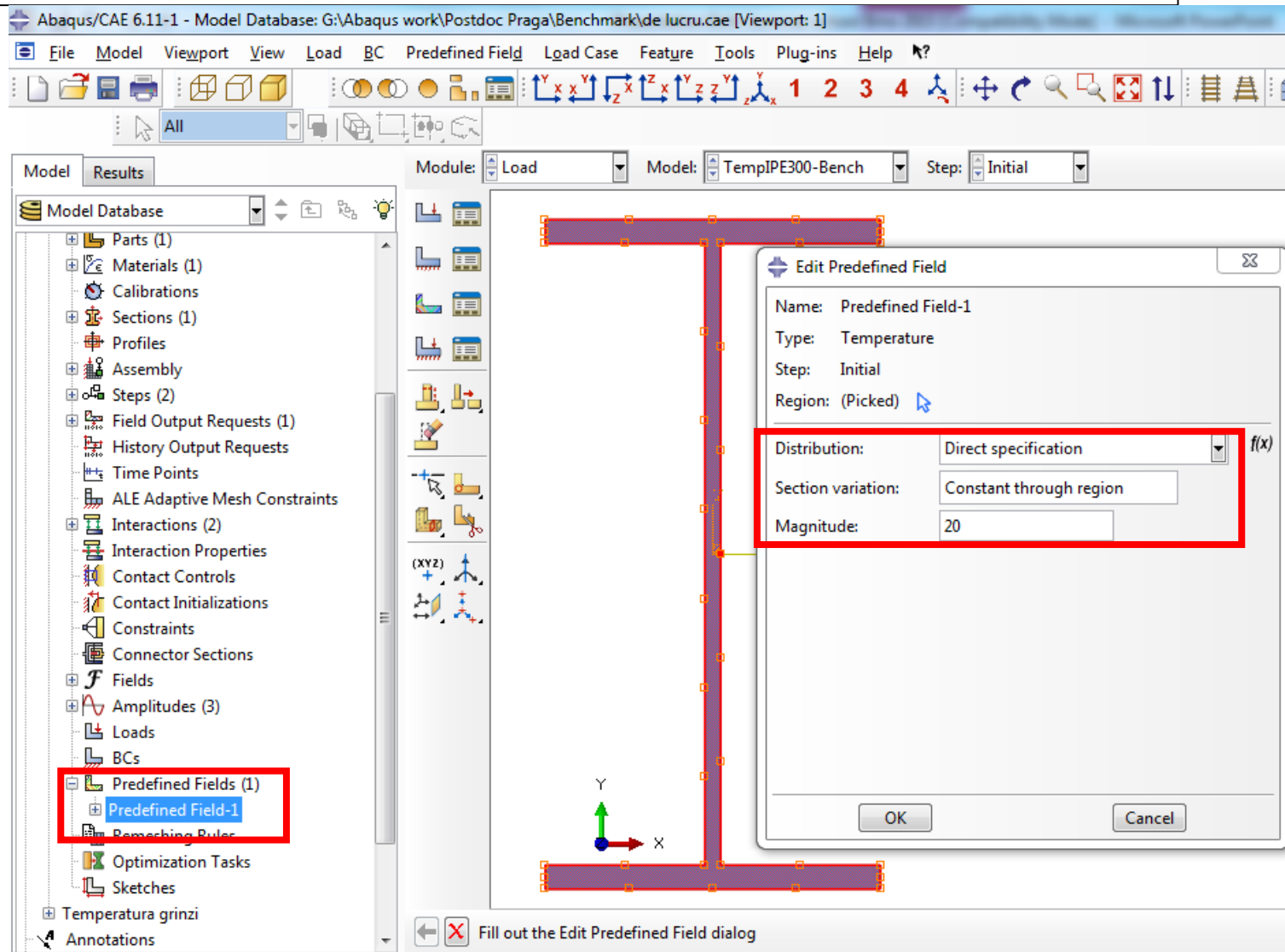
Define
mesh:
-DC2D4



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

The screenshot displays the Abaqus/CAE 6.11-1 software interface. The main window shows the 'Model Database' tree on the left, with 'Jobs (8)' expanded to show 'TempIPE300-bench (Completed)'. A context menu is open over this job, with 'Submit' highlighted in red. The 'Edit Job' dialog box is open on the right, showing the job name 'TempIPE300-bench' and the 'Submission' tab. The 'Job Type' is set to 'Full analysis', and the 'Run Mode' is set to 'Background'. The 'Submit Time' is set to 'Immediately'. The 'OK' button is visible at the bottom of the dialog.

The interface includes the following elements:

- Top Bar:** Abaqus/CAE 6.11-1 - Model Database: G:\Abaqus work\Postdoc Praga\Benchmark\de lucruc.cae [Viewport: 1]
- Menu Bar:** File, Model, Viewport, View, Job, Adaptivity, Co-execution, Optimization, Tools, Plug-ins, Help
- Toolbars:** Standard toolbar, Viewport navigation toolbar, and Job navigation toolbar.
- Model Database:** Interaction Properties, Contact Controls, Contact Initializations, Constraints, Connector Sections, Fields, Amplitudes (3), Loads, BCs, Predefined Fields (1), Remeshing Rules, Optimization Tasks, Sketches, Temperatura grinzi, Annotations, Analysis, Jobs (8), Concrete_example (Aborted), Expansion_coefficient (Completed), Gradient (Completed), Gradient-shell (Completed), Job-1 (Completed), Temp-grinzi (Completed), TempIPE300 (Completed), TempIPE300-bench (Completed), Adaptivity Processes, Co-executions, Optimization Processes.
- Edit Job Dialog:** Name: TempIPE300-bench, Model: TempIPE300-Bench, Analysis product: Abaqus/Standard, Description: [Empty], Submission tab selected, Job Type: Full analysis (selected), Recover (Explicit), Restart, Run Mode: Background (selected), Queue: [Dropdown], Host name: [Text], Type: [Text], Submit Time: Immediately (selected), Wait: [Hours] hrs. [Minutes] min., At: [Text], OK, Cancel.

Numerical model

Temperature field for secondary, unprotected beams

Save results:
- Nodal temperature

Abaqus/CAE 6.11-1 - Model Database: G:\Abaqus work\Postdoc Praga\Benchmark\de lucru.cae [Viewport: 1]

File Model Viewport View Result Plot Animate Report Options Tools Plug-ins Help

Module: Visualization ODB: G:/Abaqus work/Postdoc Praga/Benchmark/TempIPE300-b

Session Data

- Output Databases (1)
- Spectrums (7)
- XYPlots
- XYData
- Paths

NT11

+	1.048e+03
+	1.048e+03
+	1.048e+03
+	1.048e+03
+	1.048e+03
+	1.048e+03
+	1.047e+03
+	1.047e+03
+	1.047e+03
+	1.047e+03
+	1.047e+03
+	1.047e+03
+	1.047e+03
+	1.047e+03

Create XY Data

Source

- ODB history output
- ODB field output
- Thickness
- Free body
- Operate on XY data
- ASCII file
- Keyboard
- Path

Continue... Cancel

XY Data from ODB Field Output

Steps/Frames

Note: XY Data will be extracted from the active steps/frames Active Steps/Frames...

Variables Elements/Nodes

Output Variables

Position: Unique Nodal

Click checkboxes or edit the identifiers shown next to Edit below.

- HFL: Heat flux vector
- NT11: Nodal temperature
- RFL11: Reaction fluxes

Edit: NT11

Section point: All Select Settings...

Save Plot Dismiss

Y ODB: TempIPE300-bench.odb Abaqus/Standard 6.11-1

X Step: ISOfire

Increment: 1444 Step Time = 7.200

Primary Var: NT11

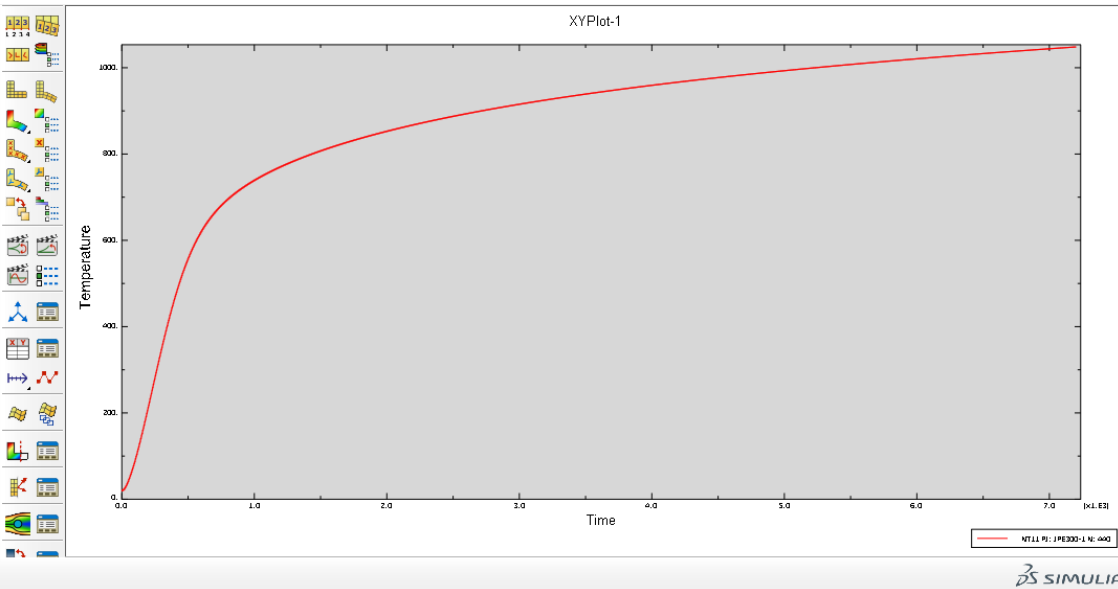
Deformed Var: not set Deformation Scale Factor: not set

Numerical model

Temperature field for secondary, unprotected beams

Save results:

- Nodal temperature



Edit XY Data

Name: _NT11 PE: 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
15	54	45.4018

Quantity Types

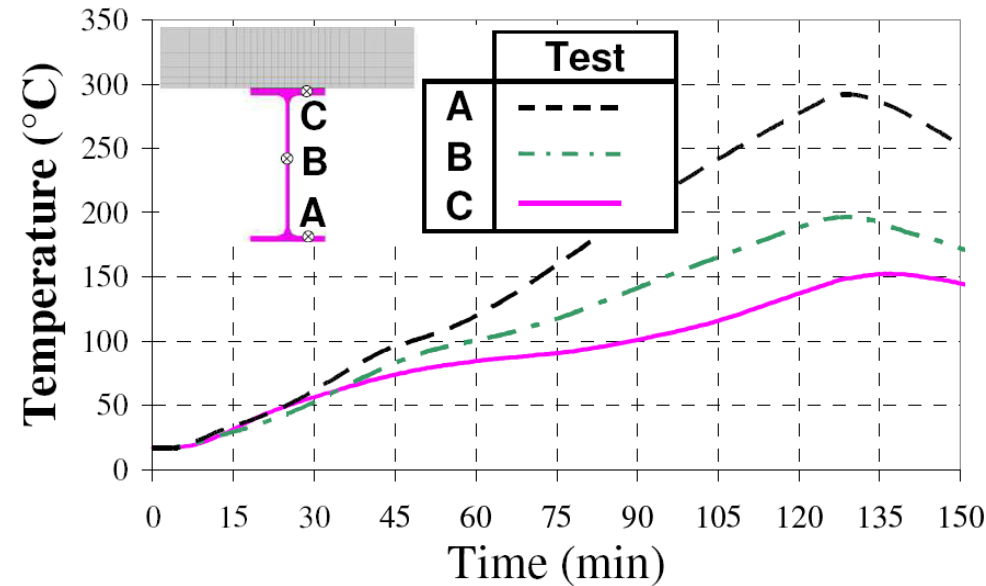
X: Time Y: Temperature

OK Cancel

Numerical model

Temperature field for protected beams

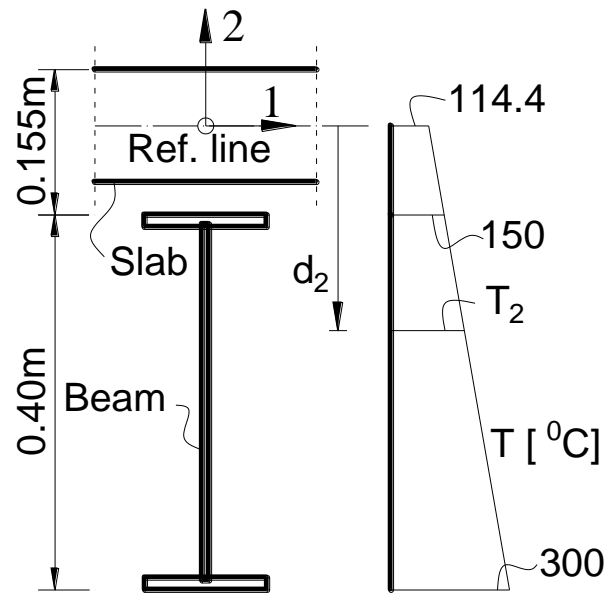
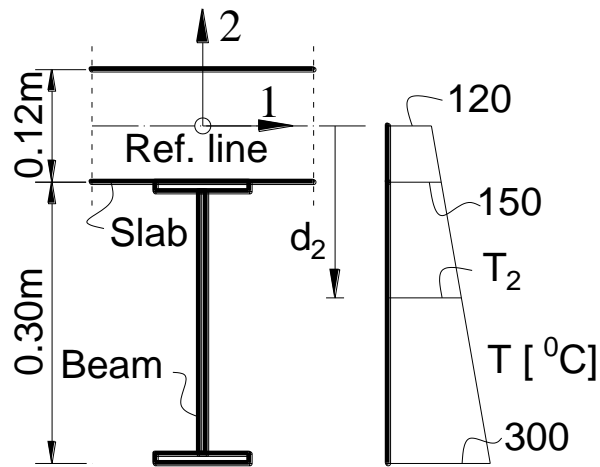
Temperature from test:
difference between top and
bottom flange



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

Temperature field for protected beams

Beam elements sections are defined function of a reference line

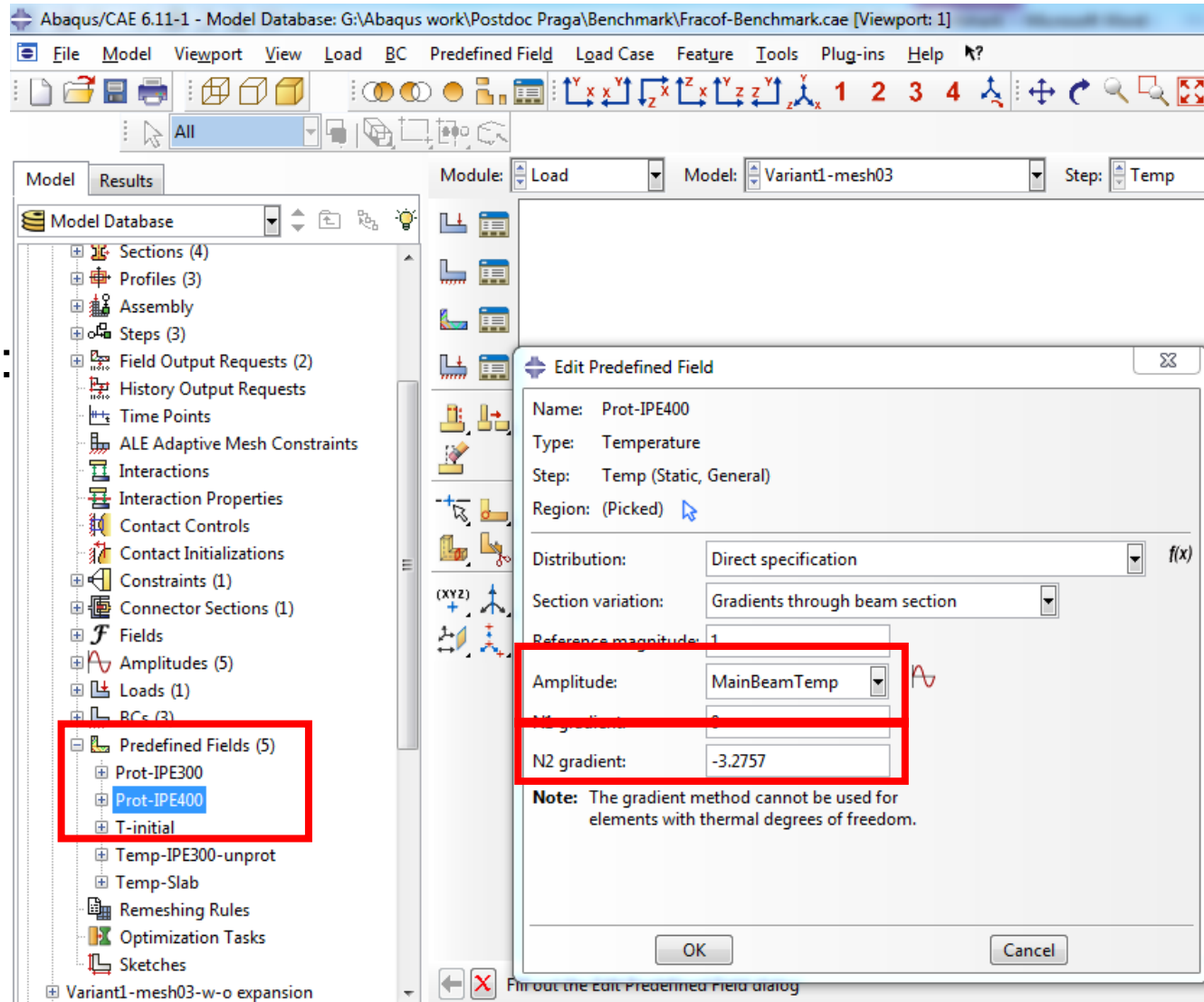


Numerical model

Temperature field for protected beams

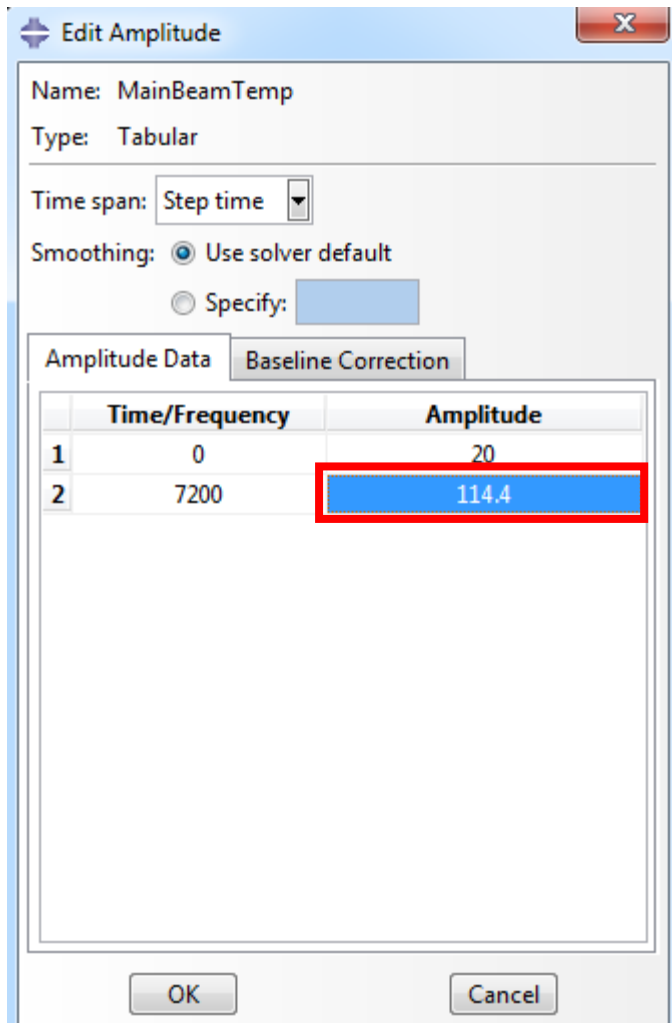
Input for *predefined field* of gradient through beam section:

- amplitude
- gradient

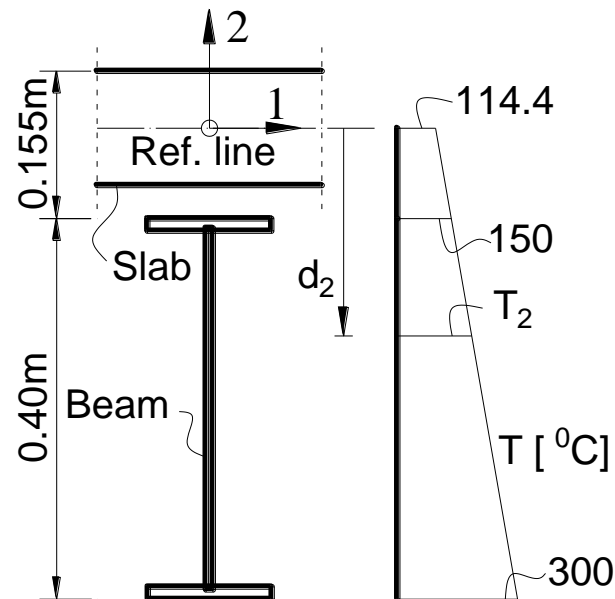


Numerical model

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

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

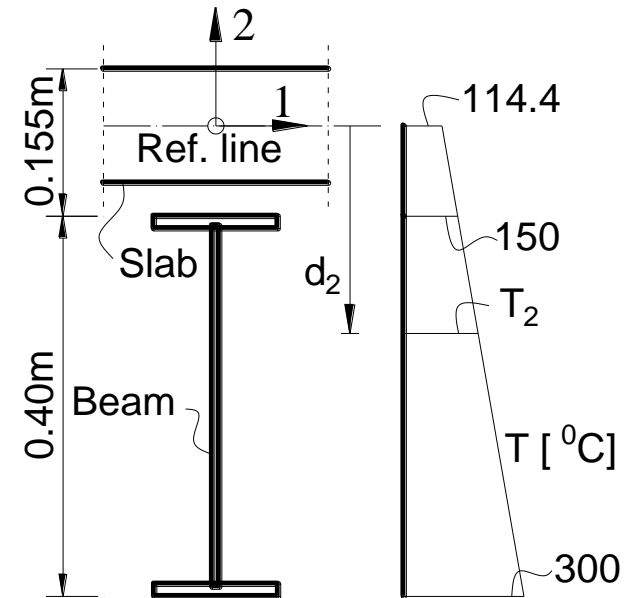
x – gradient

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

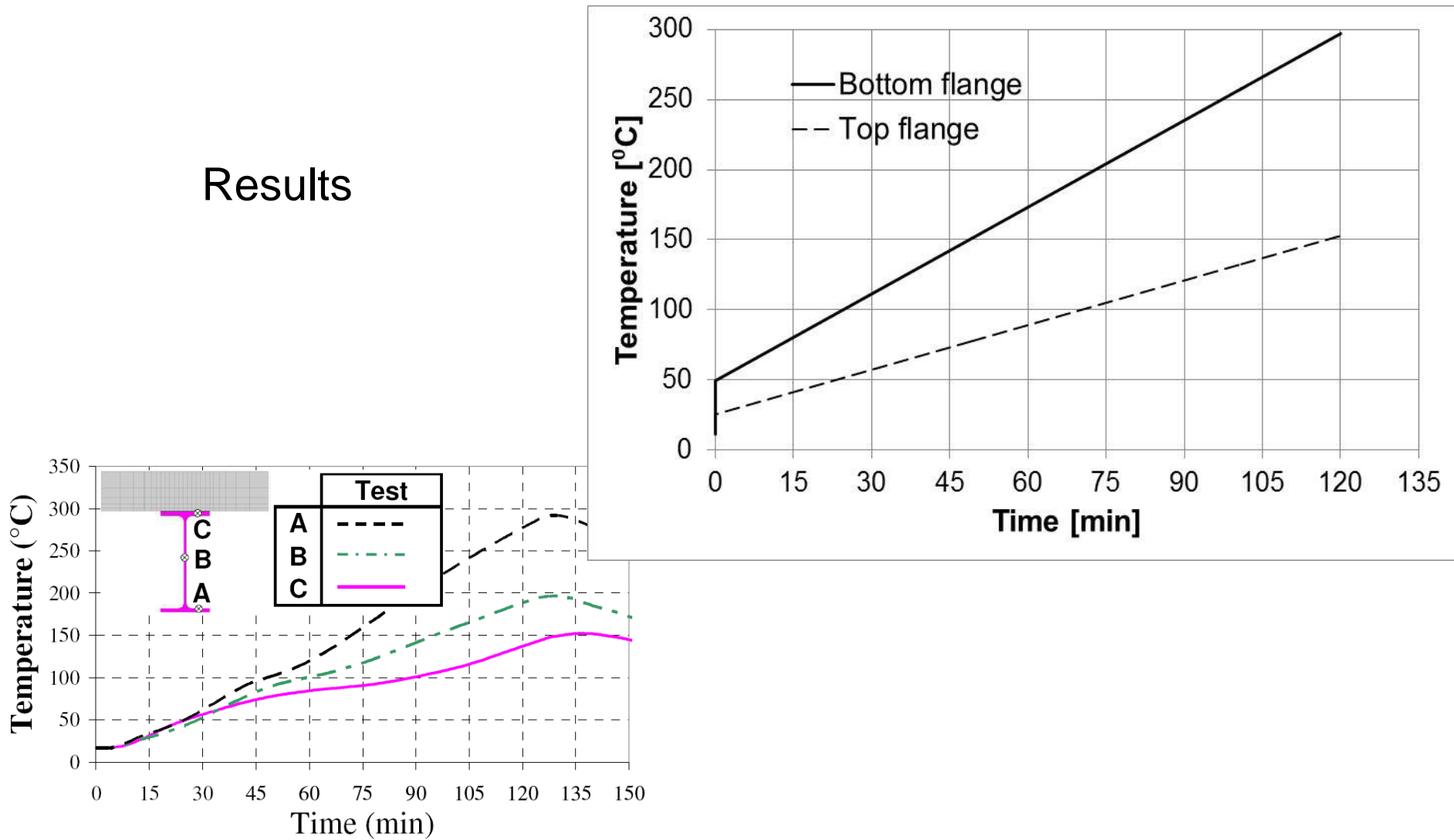
$$x \rightarrow -3.2757$$



Numerical model

Temperature field for protected beams

Results



Numerical model

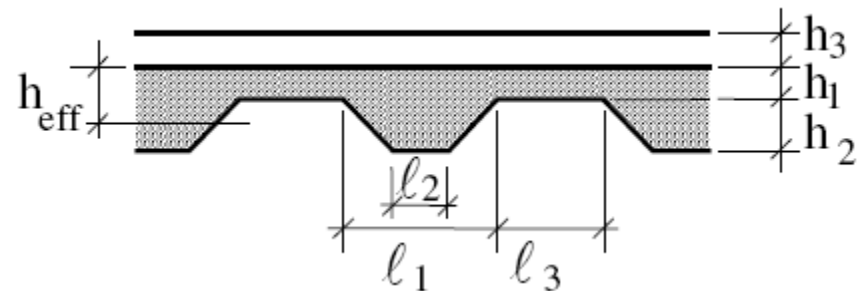
Temperature field for concrete slab

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



$$h_{eff} = h_1 + 0,5 h_2 \left(\frac{l_1 + l_2}{l_1 + l_3} \right)$$

for $h_2/h_1 \leq 1,5$ and $h_1 > 40$ mm

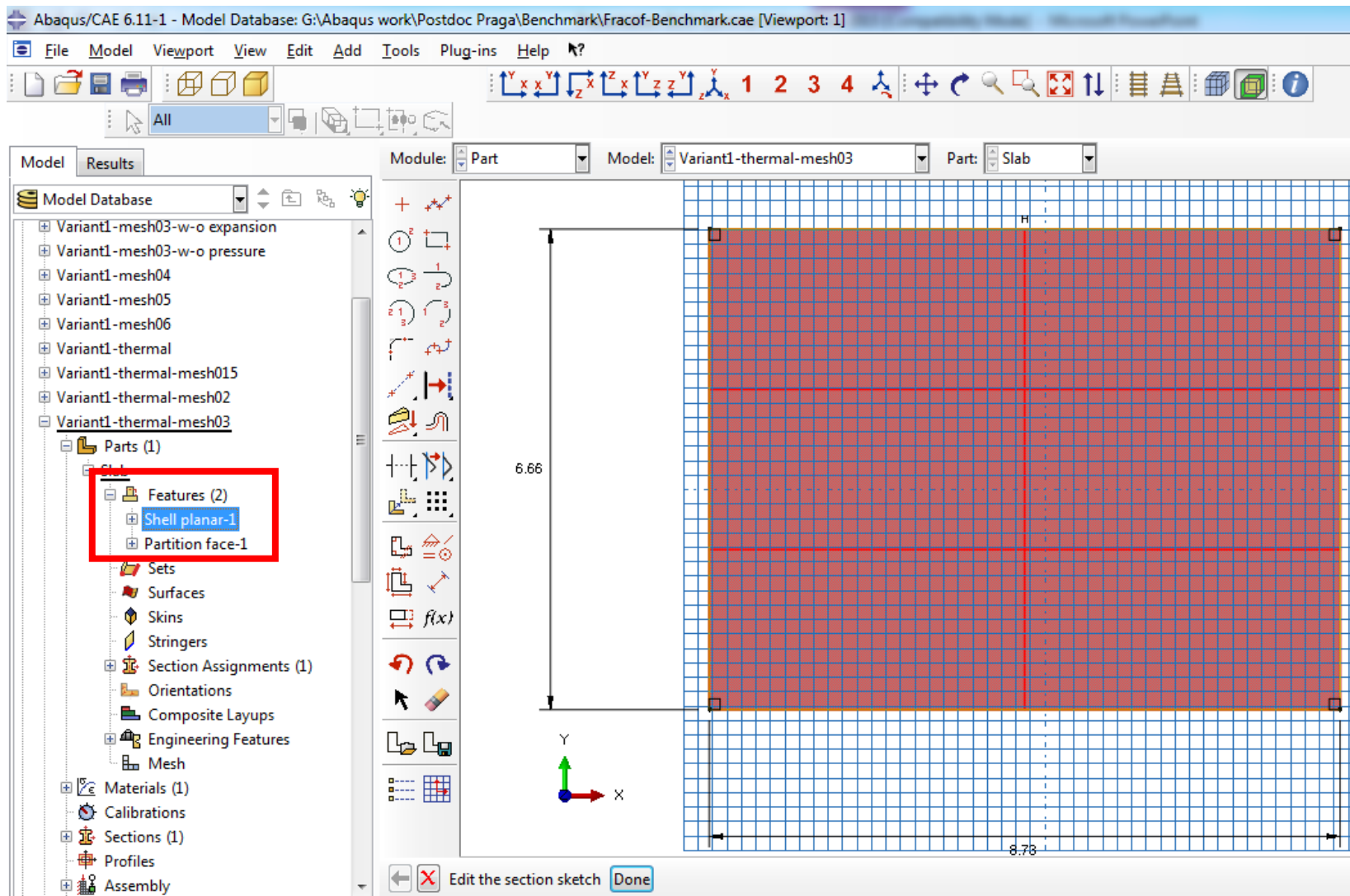
$$h_{eff} = h_1 \left[1 + 0,75 \left(\frac{l_1 + l_2}{l_1 + l_3} \right) \right]$$

for $h_2/h_1 > 1,5$ and $h_1 > 40$ mm

Numerical model

Temperature field for concrete slab

Create the part and partition:
-3D shell planar



Numerical model

Temperature field for concrete slab

Abaqus/CAE 6.11-1 - Model Database: G:\Abaqus work\Postdoc Praga\Benchmark\Fracof-Benchmark.cae [Viewport: 1]

File Model Viewport View Material Section Profile Composite Assign Special Feature Tools Plug-ins Help

Model Results Material Library

Model Database

- Variant1-mesh03-w-o expansion
- Variant1-mesh03-w-o pressure
- Variant1-mesh04
- Variant1-mesh05
- Variant1-mesh06
- Variant1-thermal
- Variant1-thermal-mesh015
- Variant1-thermal-mesh02
- Variant1-thermal-mesh03
- Parts (1)
- Materials (1)
 - Concrete
- Calibrations
- Sections (1)
- Profiles
- Assembly
- Steps (2)
- Field Output Requests (1)
- History Output Requests
- Time Points
- ALE Adaptive Mesh Constraints
- Interactions (4)
- Interaction Properties
- Contact Controls
- Contact Initializations
- Constraints
- Connector Sections
- Fields

Module: Property Model: Variant1-thermal-mesh03 Part: Slab

Edit Material

Name: Concrete

Description:

Material Behaviors

- Conductivity
- Density
- Specific Heat

General Mechanical Thermal Other

Conductivity

Type: Isotropic

Use temperature-dependent data

Number of field variables: 0

Data

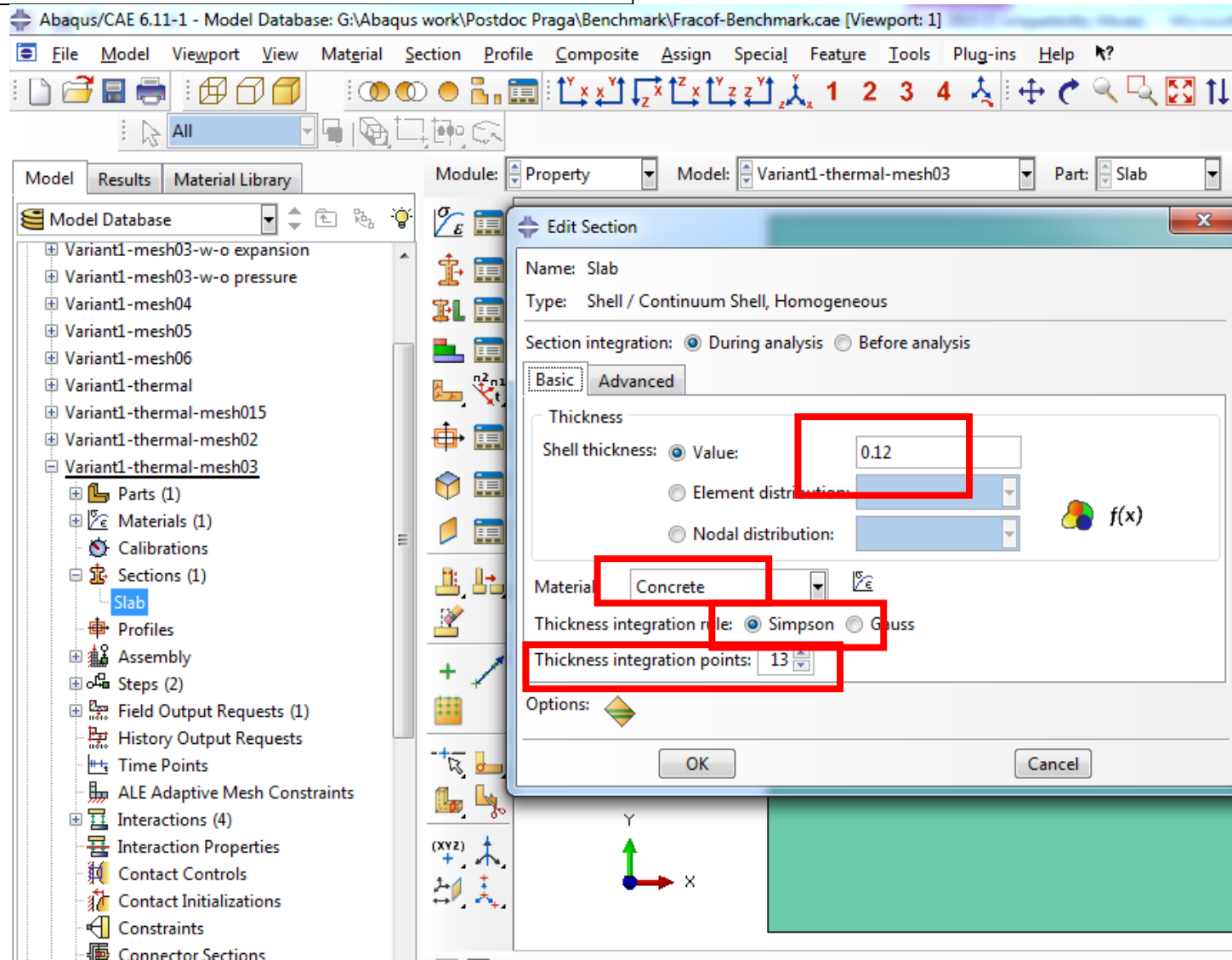
Conductivity	
1	0.9

Define material:

- conductivity
- specific heat
- density

Numerical model

Temperature field for concrete slab



Define section:

-thickness

-material

-integration
rule

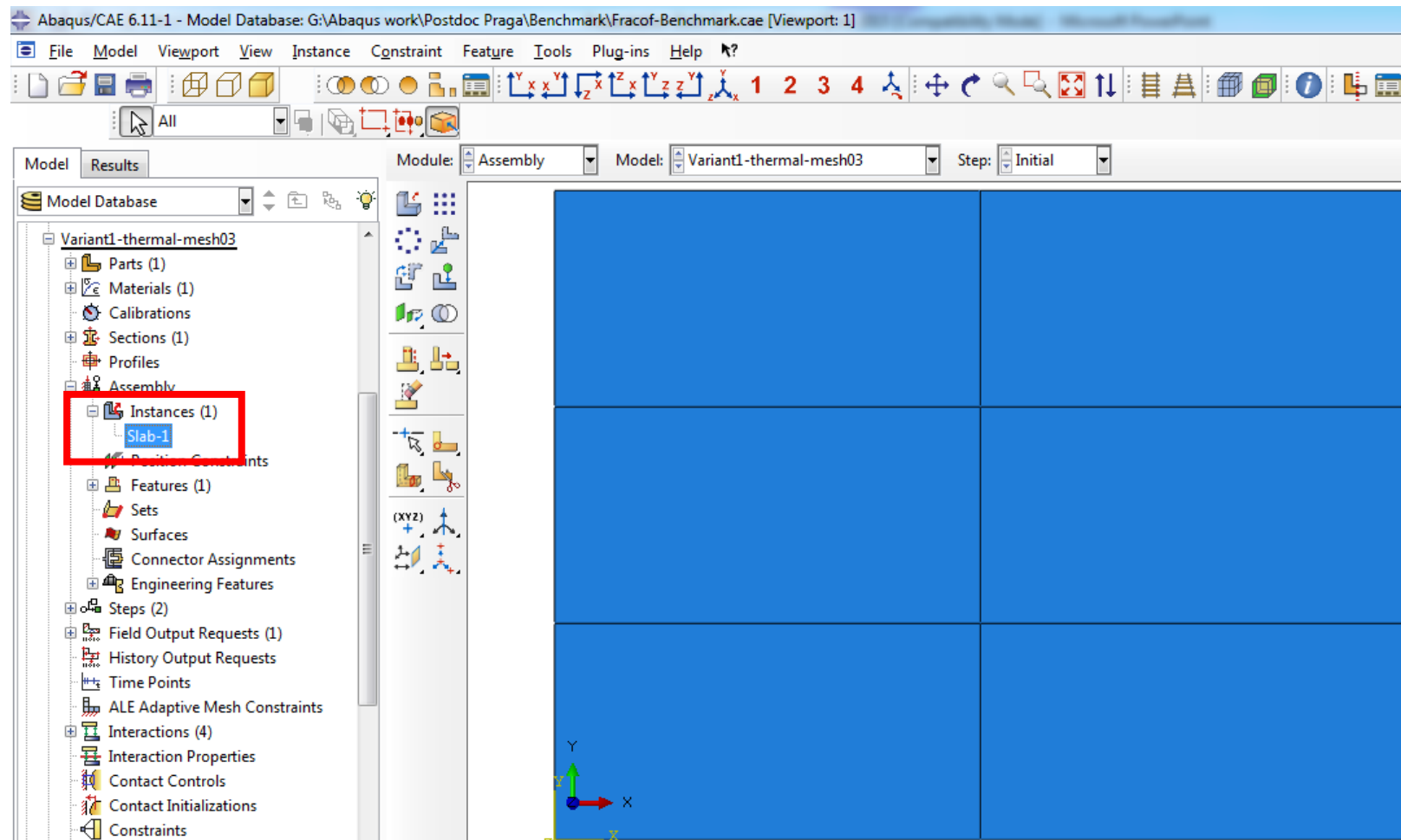
-integration
points

(without
reinforcement, yet)

Numerical model

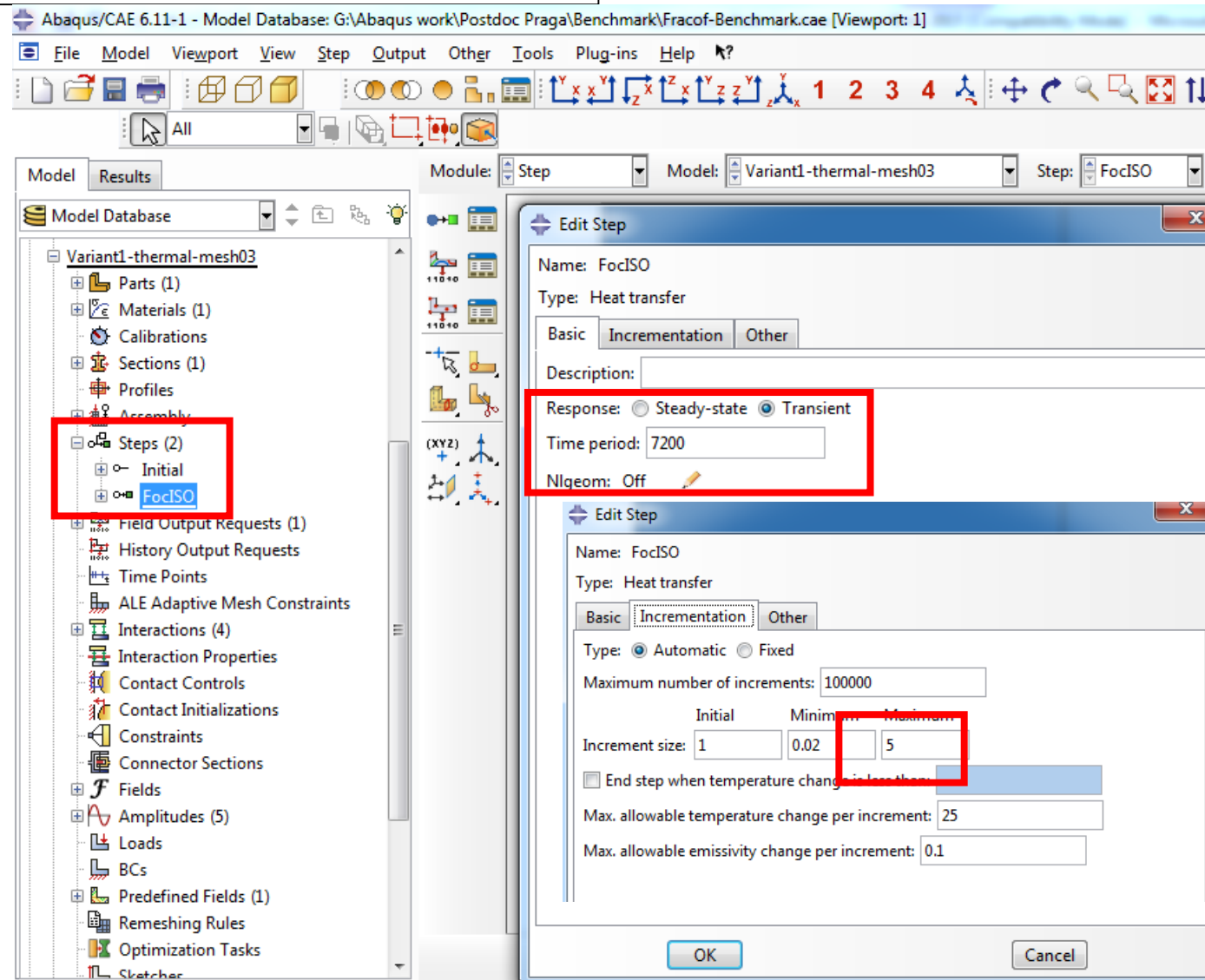
Temperature field for concrete slab

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

Edit Interaction

Name: Conv-heated

Type: Surface film condition

Step: FocISO (Heat transfer)

Surface: (Picked)

Definition: Embedded Coefficient

Film coefficient: 25

Film coefficient amplitude: (Instantaneous)

Sink definition: Uniform

Sink temperature: 1

Sink amplitude: Foc-ISO-C

Edit Interaction

Name: Conv-unheated

Type: Surface film condition

Step: FocISO (Heat transfer)

Surface: (Picked)

Definition: Embedded Coefficient

Film coefficient: 9

Film coefficient amplitude: (Instantaneous)

Sink definition: Uniform

Sink temperature: 20

Sink amplitude: (Instantaneous)

OK Cancel

Edit Interaction

Name: Rad-heated

Type: Surface radiation

Step: FocISO (Heat transfer)

Surface: (Picked)

Radiation type: To ambient Cavity approximation (3D only)

Emissivity distribution: Uniform

Emissivity: 0.7

Ambient temperature: 1

Ambient temperature amplitude: Foc-ISO-C

OK Cancel

Edit Interaction

Name: Rad-unheated

Type: Surface radiation

Step: FocISO (Heat transfer)

Surface: (Picked)

Radiation type: To ambient Cavity approximation (3D only)

Emissivity distribution: Uniform

Emissivity: 0.7

Ambient temperature: 20

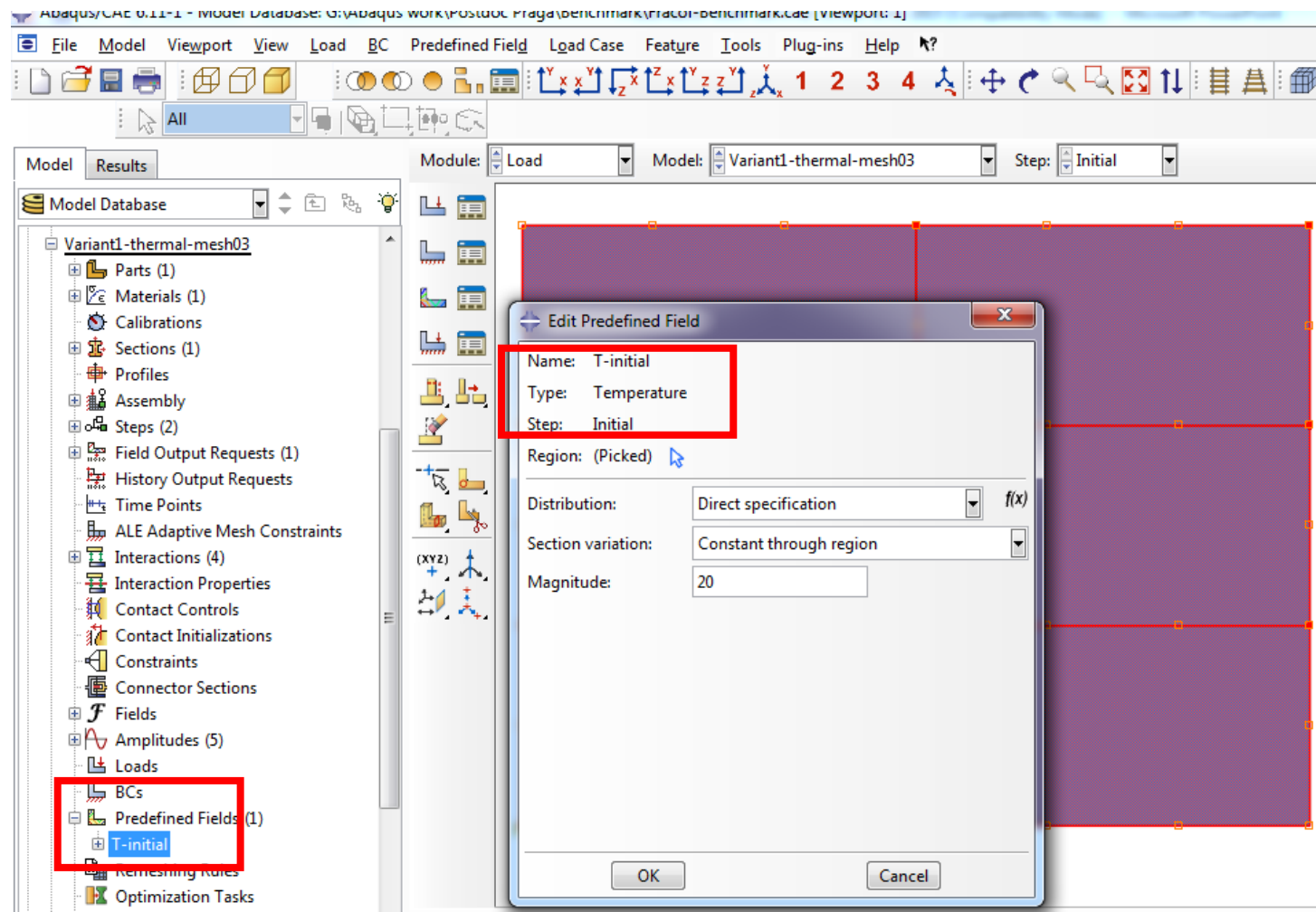
Ambient temperature amplitude: (Instantaneous)

OK Cancel

Numerical model

Temperature field for concrete slab

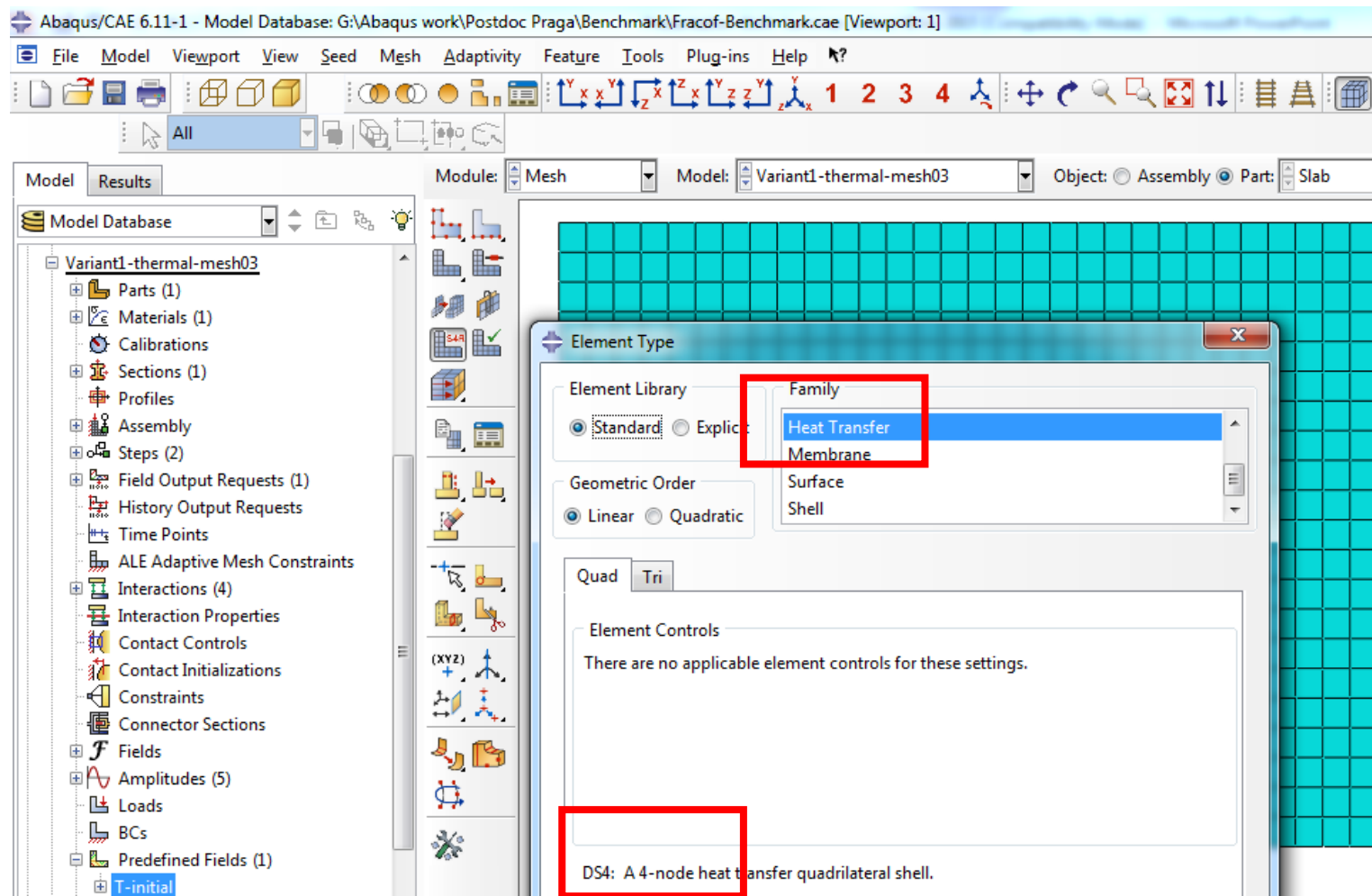
Define initial temperature:
-predefined field – constant through region



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

The screenshot displays the Abaqus/CAE software interface. The main window shows the 'Edit Job' dialog box for a job named 'Variant1-thermal-mesh03'. The 'Submission' tab is active, showing options for 'Job Type' (Full analysis selected) and 'Run Mode' (Background selected). The 'Number of domains' is set to 3. The 'Parallelization' tab is also visible, showing 'Use multiple processors' checked and 'Number of processors' set to 3. A list of jobs is shown in the background, with 'Variant1-thermal-mesh03 (Completed)' highlighted. The 'Jobs (16)' folder is highlighted with a red box, and the 'Parallelization' tab in the 'Edit Job' dialog is also highlighted with a red box.

Abaqus/CAE 6.11-1 - Model Database: G:\Abaqus work\Postdoc Praga\Benchmark\Fracof-Benchmark.cae [Viewport: 1]

File Model Viewport View Job Adaptivity Co-execution Optimization Tools Plug-ins Help

Model: Variant1-thermal-mesh03 Step:

Model Database

- Optimization Tasks
- Sketches
- Variant1-thermal-mesh04
- Variant1-thermal-mesh05
- Variant1-thermal-mesh06
- Annotations
- Analysis
- Jobs (16)
 - Variant1 (Aborted)
 - Variant1-mesh01 (Completed)
 - Variant1-mesh02 (Completed)
 - Variant1-mesh03 (Completed)
 - Variant1-mesh03-faraExpans (Completed)
 - Variant1-mesh03-faraPressure (Completed)
 - Variant1-mesh04 (Completed)
 - Variant1-mesh05 (Completed)
 - Variant1-mesh06 (Completed)
 - Variant1-thermal (Completed)
 - Variant1-thermal-mesh015 (Completed)
 - Variant1-thermal-mesh02 (Completed)
 - Variant1-thermal-mesh03 (Completed)**
 - Variant1-thermal-mesh04 (Completed)
 - Variant1-thermal-mesh05 (Completed)
 - Variant1-thermal-mesh06 (Completed)
- Adaptivity Processes
- Co-executions
- Optimization Processes

Name: Variant1-thermal-mesh03
Model: Variant1-thermal-mesh03
Analysis product: Abaqus/Standard

Description:

Submission General Memory Parallelization Precision

Use multiple processors 3

Abaqus/Explicit

Number of domains: 3

Parallelization method: Domain

Multiprocessing mode: Default

Edit Job

Name: Variant1-thermal-mesh03
Model: Variant1-thermal-mesh03
Analysis product: Abaqus/Standard

Description:

Submission General Memory Parallelization Precision

Job Type

Full analysis
 Recover (Explicit)
 Restart

Run Mode

Background Queue: Host name: Type:

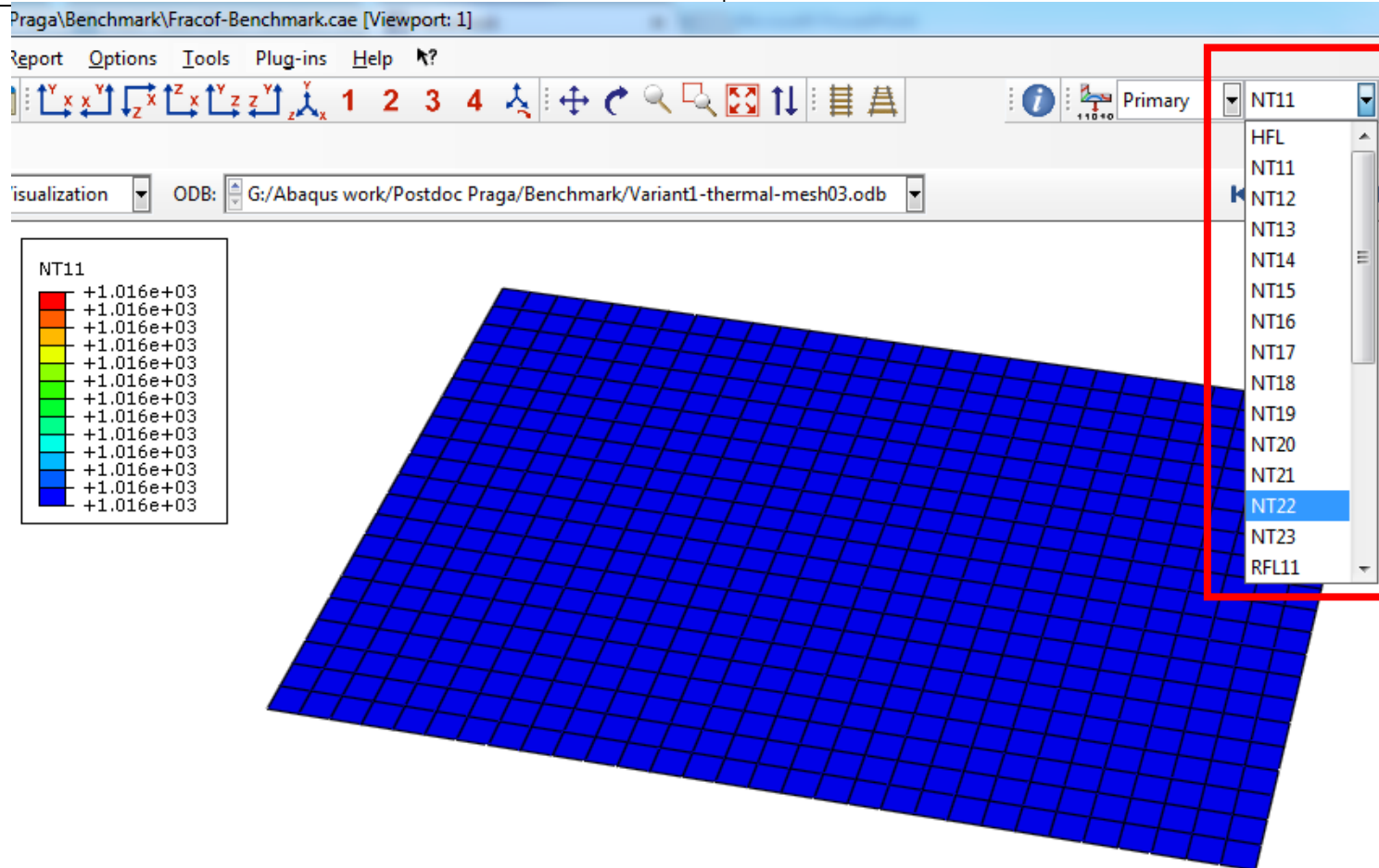
Submit Time

Immediately
 Wait: hrs. min.
 At:

Numerical model

Temperature field for concrete slab

Results:
Nodal
temperatures



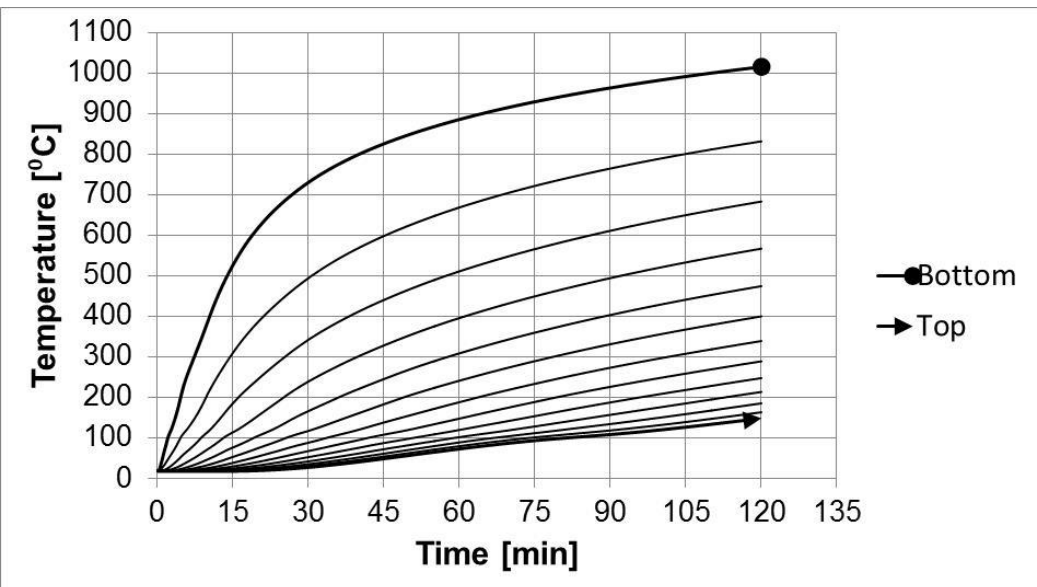
ODB: Variant1-thermal-mesh03.odb Abaqus/Standard 6.11-1 Sun Nov 02 12:08:48 GTB Standard Time 2014

Step: FocISO
Increment: 1444; Step Time = 7200.
Primary Var: NT11
Deformed Var: not set; Deformation Scale Factor: not set

Numerical model

Temperature field for concrete slab

Results:
Nodal
temperatures



XY Data from ODB Field Output

Steps/Frames
Note: XY Data will be extracted from the active steps/frames [Active Steps/Frames...](#)

Variables **Elements/Nodes**

Output Variables
 Position: Unique Nodal

Click checkboxes or edit the identifiers shown next to Edit below.

- HFL: Heat flux vector
- NT11: Nodal temperature
- NT12: Nodal temperature
- NT13: Nodal temperature
- NT14: Nodal temperature
- NT15: Nodal temperature
- NT16: Nodal temperature

Edit: NT11,NT12,NT13,NT14,NT15,NT16

Section point: All Select [Settings...](#)

Save Plot Dismiss

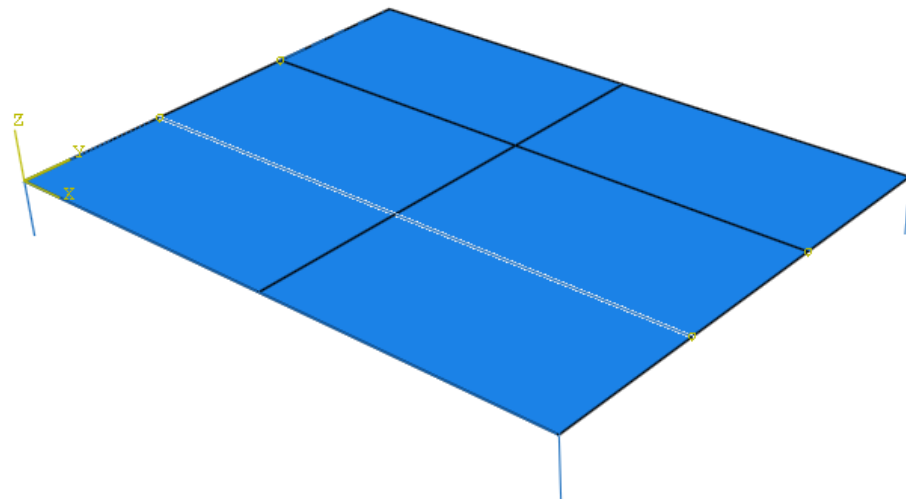
Numerical model

Structural analysis of composite slab

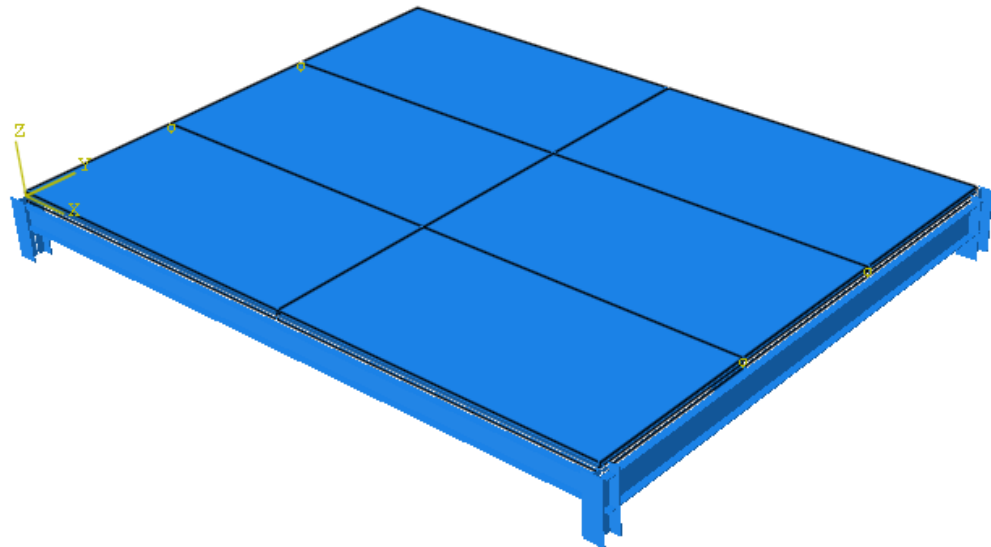
Starts from a saved as model of thermal analysis of concrete slab

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

Abaqus/CAE 6.11-1 - Model Database: G:\Abaqus work\Postdoc Praga\Benchmark\Fracof-Benchmark.cae [Viewport: 1]

File Model Viewport View Material Section Profile Composite Assign Special Feature Tools Plug-ins Help

Model Results Material Library

Model Database

- Variant1-mesh03
 - Parts (4)
 - HEB260
 - IPE300
 - IPE400
 - Slab
 - Materials (4)
 - Calibrations
 - Sections (4)
 - HEB260
 - IPE300
 - IPE400
 - Slab
 - Profiles (3)
 - Assembly
 - Steps (3)
 - Field Output Requests (2)
 - History Output Requests
 - Time Points
 - ALE Adaptive Mesh Constraints
 - Interactions
 - Interaction Properties
 - Contact Controls
 - Contact Initializations
 - Constraints (1)
 - Connector Sections (1)
 - Fields

Module: Edit Section

Name: Slab
Type: Shell / Continuum Shell, Homogeneous
Section integration: During analysis Before analysis

Basic Advanced

Thickness

Shell thickness: Value: 0.12
 Element distribution:
 Nodal distribution: $f(x)$

Material: Concrete ϵ

Thickness integration rule: Simpson Gauss

Thickness integration points: 13

Options:

OK Cancel

Rebar Layers

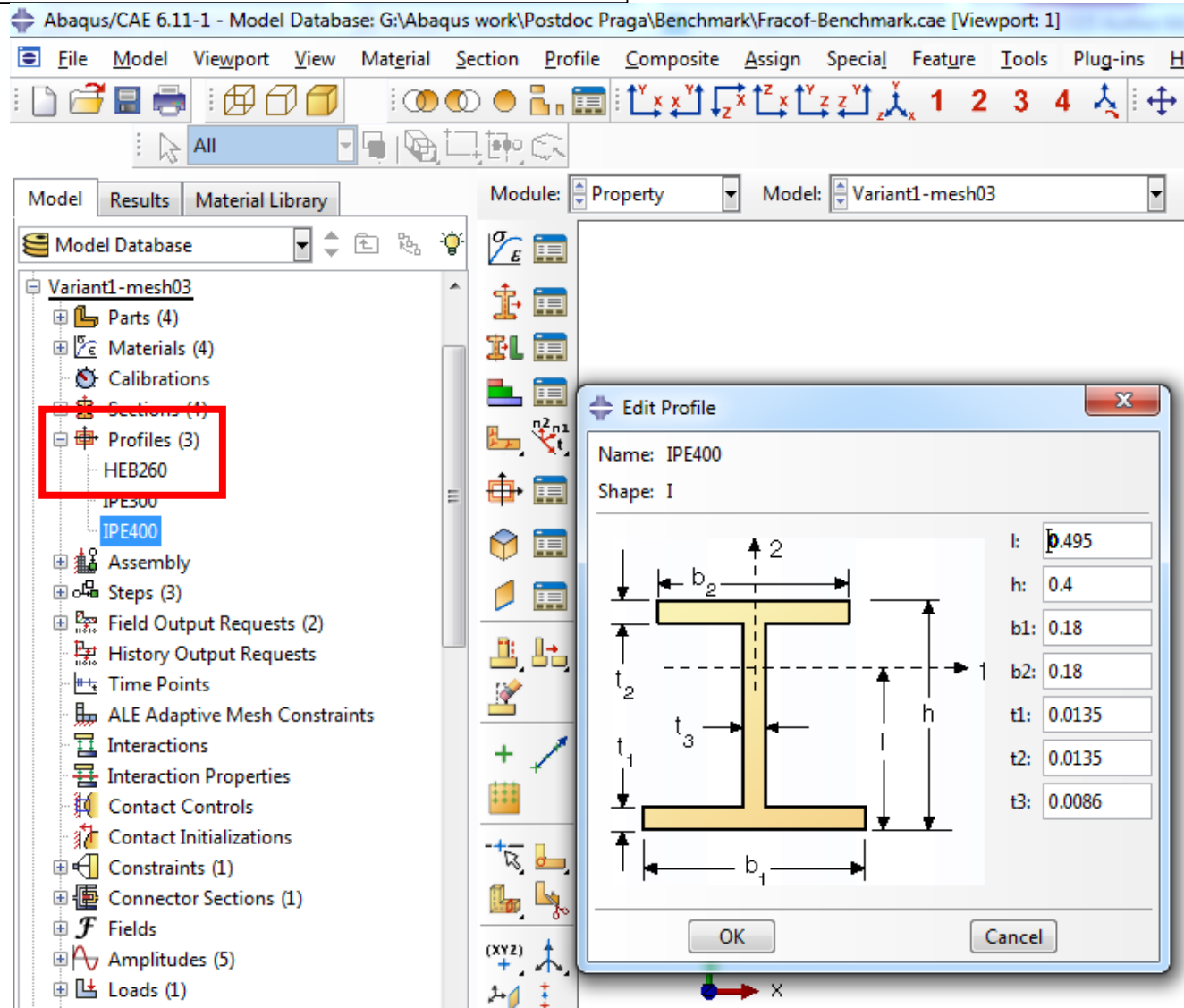
Rebar geometry: Constant Angular Lift equation-based

Layer Name	Material	Area per Bar	Spacing	Orientation Angle	Position
Long	S500-rebar	3.8484E-005	0.15	0	0.01
Trans	S500-rebar	3.8484E-005	0.15	90	0.01

Numerical model

Structural analysis of composite slab

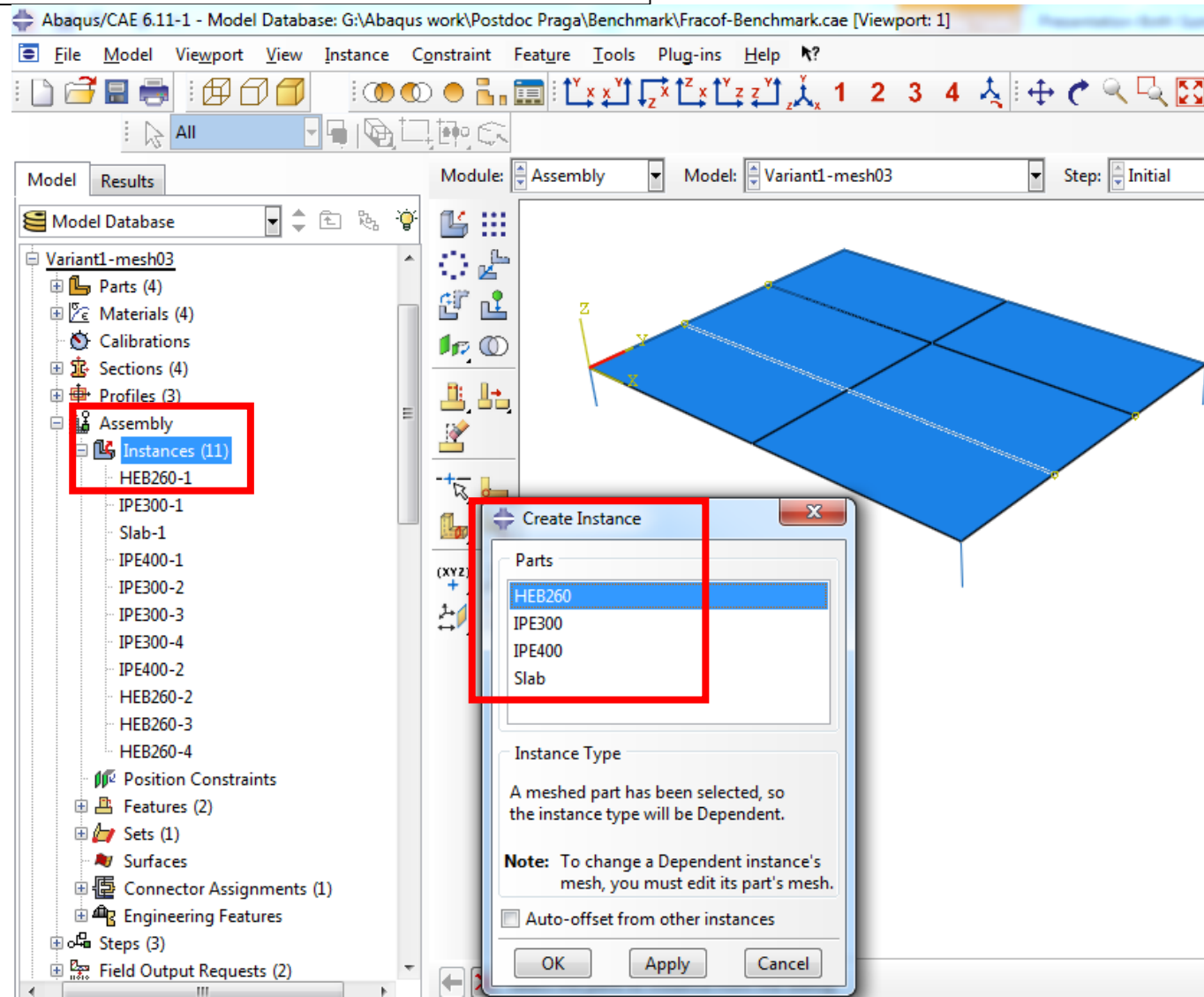
Define “profiles”
for the wire
elements and
orientation



Numerical model

Structural analysis of composite slab

Create
“instances” from
parts for all
elements and
“construct” the
structure



Numerical model

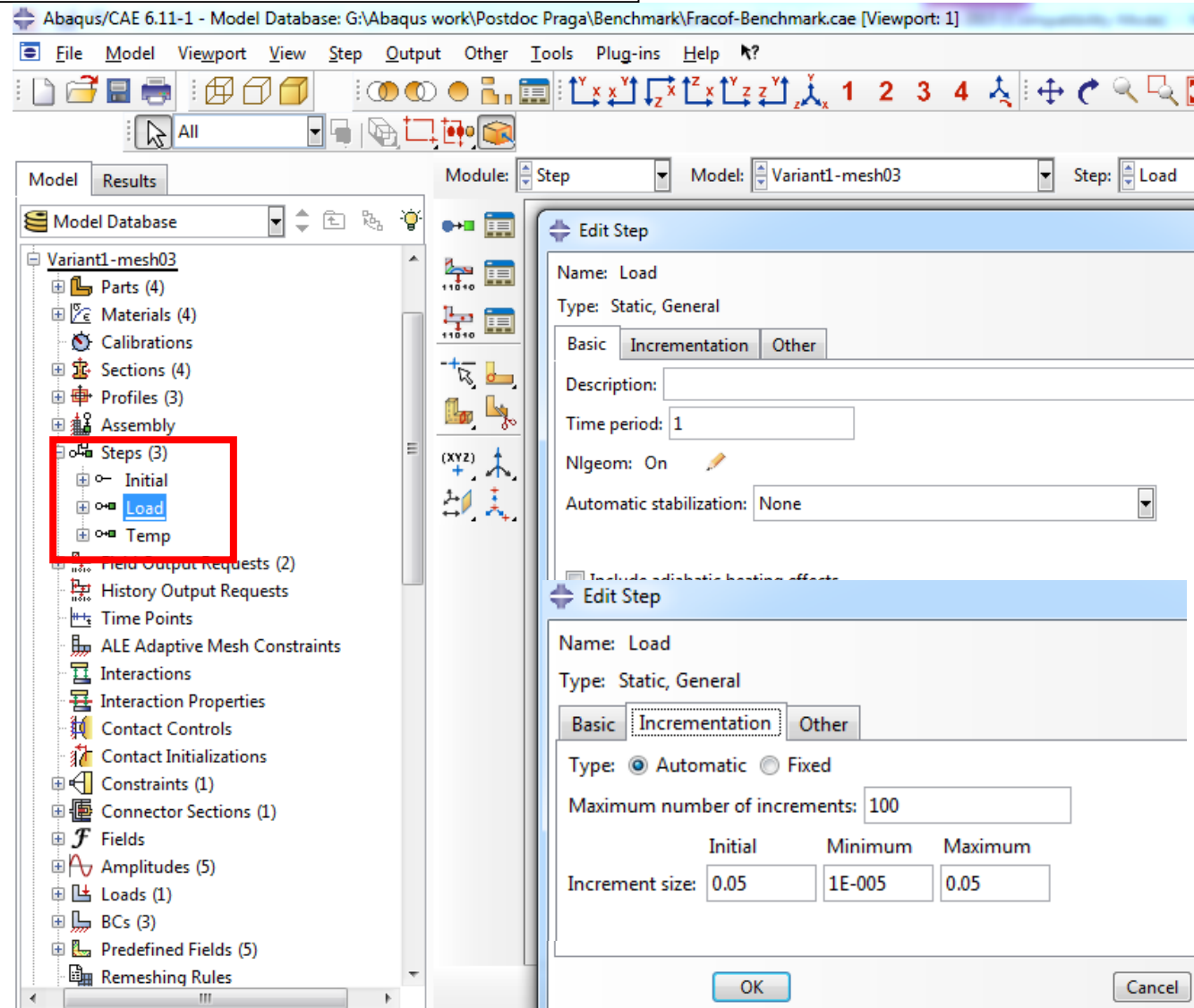
Structural analysis of composite slab

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

The screenshot displays the Abaqus/CAE 6.11-1 software interface. The title bar indicates the model database is located at G:\Abaqus work\Postdoc Praga\Benchmark\Fracof-Benchmark.cae. The main window shows the Model Database tree on the left, where the 'Temp' step is highlighted in blue. The 'Edit Step' dialog box is open on the right, showing the configuration for the 'Temp' step. The dialog is set to 'Static, General' type and 'Automatic' type. The 'Maximum number of increments' is set to 100000, and the 'Increment size' is set to 0.05, 1E-005, and 5 for Initial, Minimum, and Maximum increments, respectively.

Model Database Tree:

- Variant1-mesh03
 - Parts (4)
 - Materials (4)
 - Calibrations
 - Sections (4)
 - Profiles (3)
 - Assembly
 - Steps (3)
 - Initial
 - Load
 - Temp**
 - Field Output Requests (2)
 - History Output Requests
 - Time Points
 - ALE Adaptive Mesh Constraints
 - Interactions
 - Interaction Properties
 - Contact Controls
 - Contact Initializations
 - Constraints (1)
 - Connector Sections (1)
 - Fields
 - Amplitudes (5)
 - Loads (1)
 - BCs (3)
 - Predefined Fields (5)
 - Remeshing Rules

Edit Step Dialog:

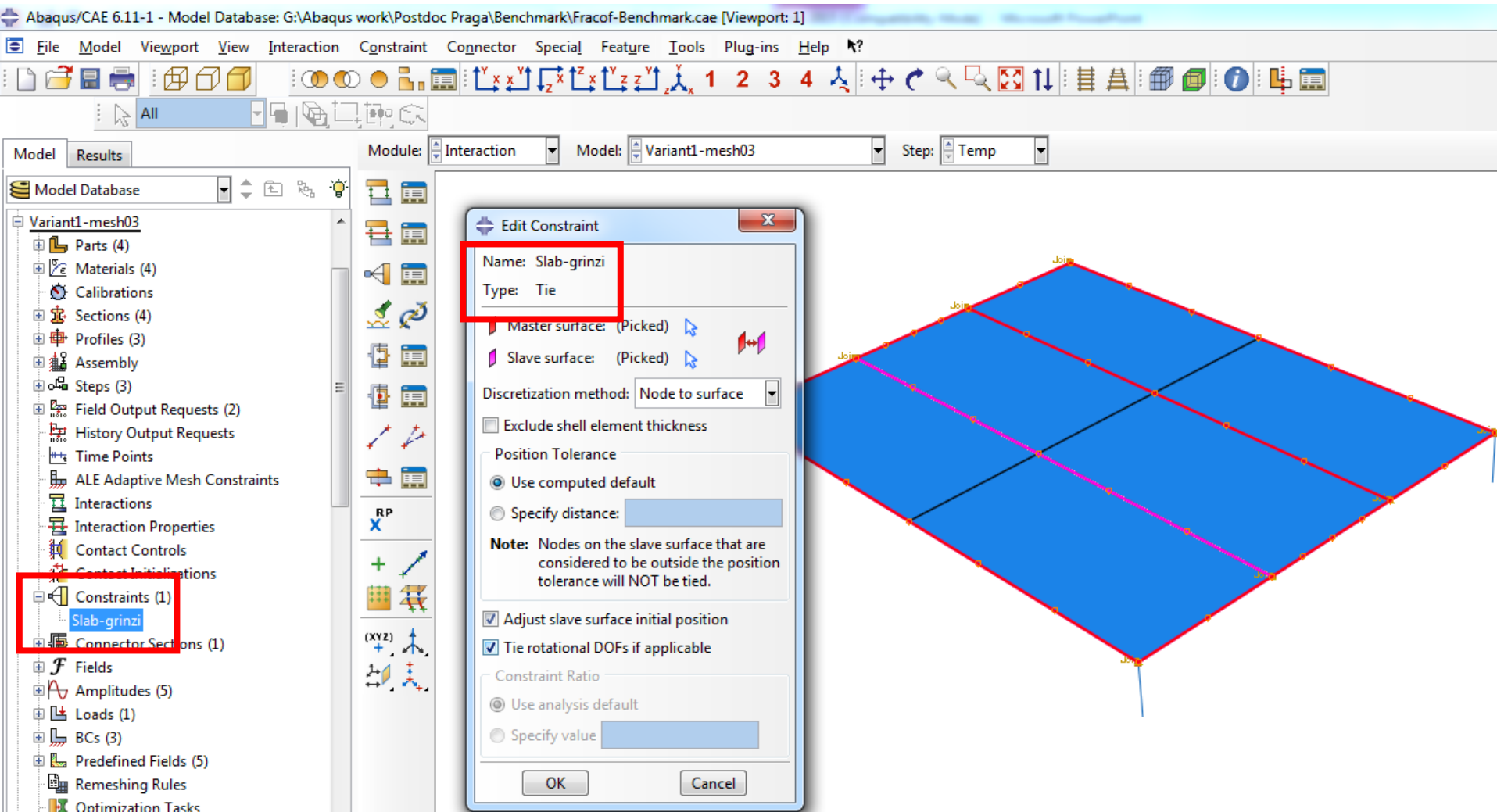
- Name: Temp
- Type: Static, General
- Basic | Incrementation | Other
- Description:
- Time period: 7200
- Nlgeom: On
- Automatic stabilization: None
- Name: Temp
- Type: Static, General
- Basic | Incrementation | Other
- Type: Automatic Fixed
- Maximum number of increments: 100000
- Increment size:

	Initial	Minimum	Maximum
Increment size:	0.05	1E-005	5

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"*

Abaqus/CAE 6.11-1 - Model Database: G:\Abaqus work\Postdoc Praga\Benchmark\Fracof-Benchmark.cae [Viewport: 1]

File Model Viewport View Interaction Constraint Connector Special Feature Tools Plug-ins Help

All

Model Results

Model Database

- Variant1-mesh03
 - Parts (4)
 - Materials (4)
 - Calibrations
 - Sections (4)
 - Profiles (3)
 - Assembly
 - Steps (3)
 - Field Output Requests (2)
 - History Output Requests
 - Time Points
 - ALE Adaptive Mesh Constraints
 - Interactions
 - Interaction Properties
 - Contact Controls
 - Contact Initializations
 - Constraints (1)
 - Connector Sections (1)
 - SectioniuneConnector
 - Fields
 - Amplitudes (5)
 - Loads (1)
 - BCs (3)
 - Predefined Fields (5)
 - Remeshing Rules
 - Optimization Tasks
 - Sketches

Module:

Edit Connector Section

Name: SectioniuneConnector

Type: Join

Available CORM: None Constrained CORM: U1, U2, U3

Connection type diagram:

Behavior Options Table Options Section Data

Table Options

Regularization

Regularize data (Explicit only)

Tolerance:

Use default Specify: 0.03

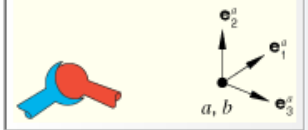
Extrapolation

Constant Linear

Note: These settings apply to all tabular data specified within this connector section, unless superseded by settings specified for an individual table.

Connection Type Tip

Translational type: Join



Dismiss

Edit Connector Section Assignment

Region: wireconnectors

Section Orientation 1 Orientation 2

Section: SectioniuneConnector

Connection type: Join

Connection type diagram:

OK Cancel

Numerical model

Structural analysis of composite slab

Define *Amplitudes* :

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

The screenshot shows the Abaqus/CAE 6.11-1 software interface. The 'Model Database' tree on the left has 'Amplitudes (5)' expanded, with 'TempUnprotected' selected and highlighted in a red box. The 'Edit Amplitude' dialog box is open, showing the following details:

- Name: TempUnprotected
- Type: Tabular
- Time span: Step time
- Smoothing: Use solver default
- Specify: []

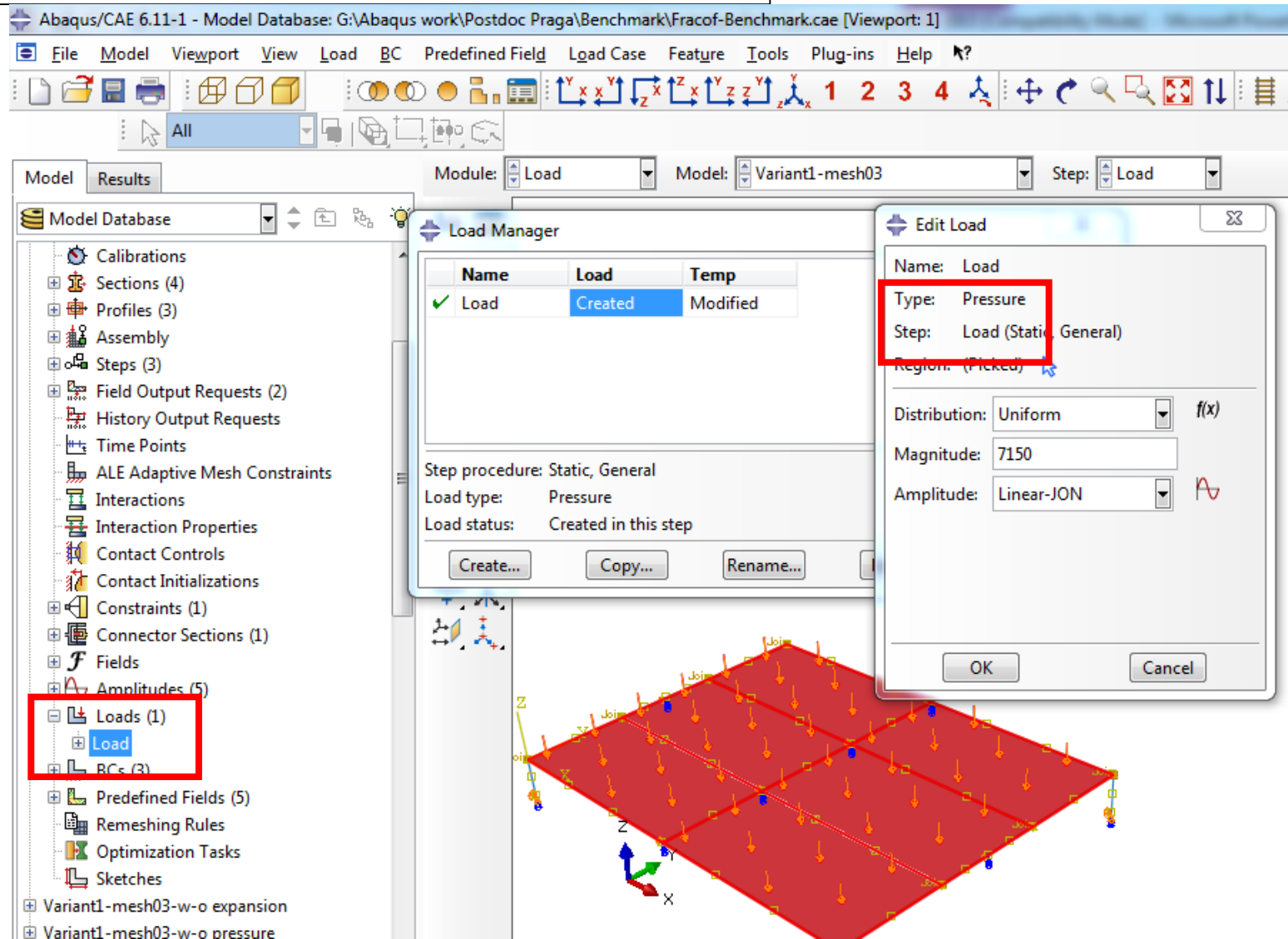
The 'Amplitude Data' tab is active, displaying a table with the following data:

	Time/Frequency	Amplitude
1	0	20
2	9	20.9886
3	29	30.288
4	49	46.2808
5	69	65.9193
6	89	87.4783
7	109	110.774
8	149	161.176
9	179	200.295
10	199	226.598
11	249	291.391
12	299	352.905
13	349	419.774

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)

The image shows the Abaqus software interface. The main window displays a 3D model of a composite slab with a mesh. Two dialog boxes are open:

Predefined Field Manager

Name	Initial	Load	Temp
✓ Prot-IPE300			Created
✓ Prot-IPE400			Created
✓ T-initial	Created	Propagated	Propagated
✓ Temp-IPE300-			Created
✓ Temp-Slab			Created

Edit Predefined Field

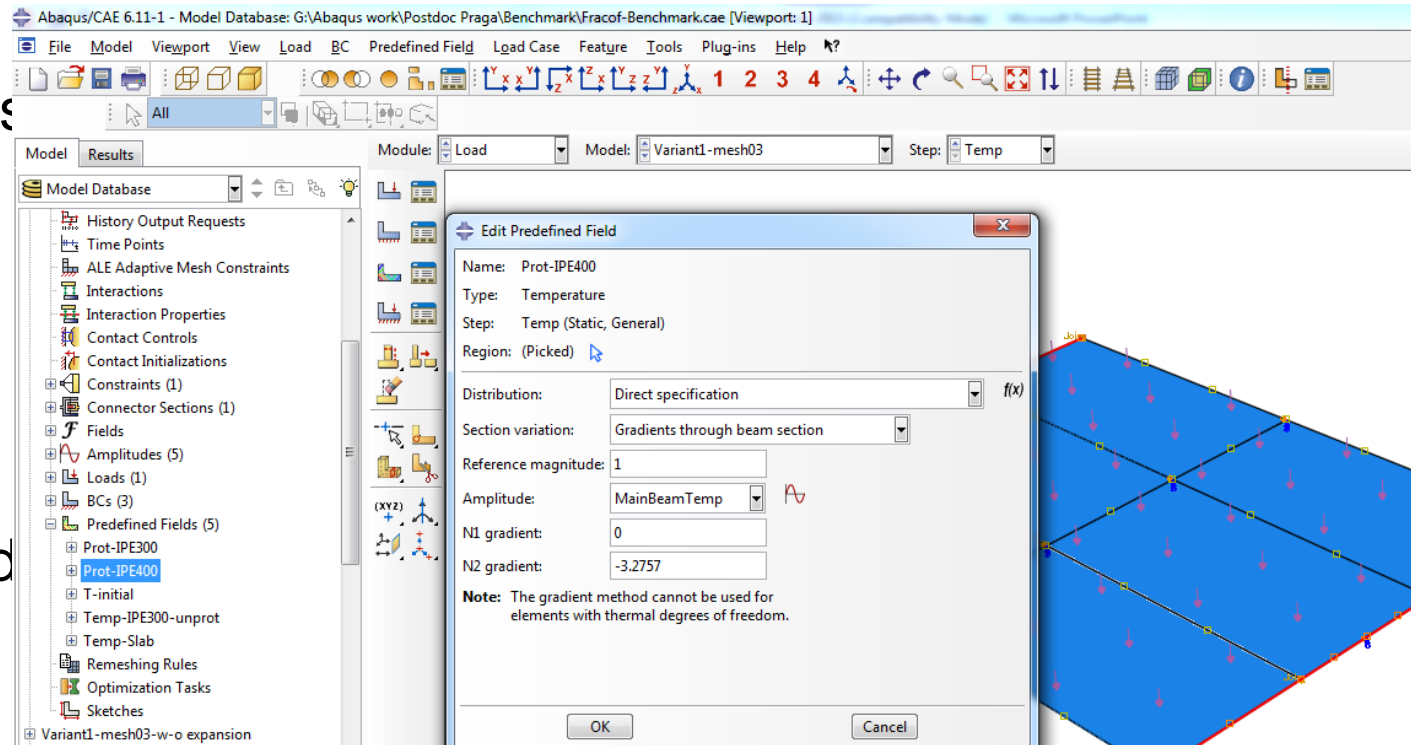
Name: T-initial
 Type: Temperature
 Step: Temp (Static, General)
 Region: (Picked)
 Status: Propagated
 Distribution: Direct specification
 Section variation: Constant through region
 Magnitude: 20
 Amplitude: (Ramp)

The background shows the Abaqus Model Database tree with 'Predefined Fields (5)' expanded, listing 'Prot-IPE300', 'Prot-IPE400', 'T-initial', 'Temp-IPE300-unprot', and 'Temp-Slab'. The 'T-initial' field is selected.

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

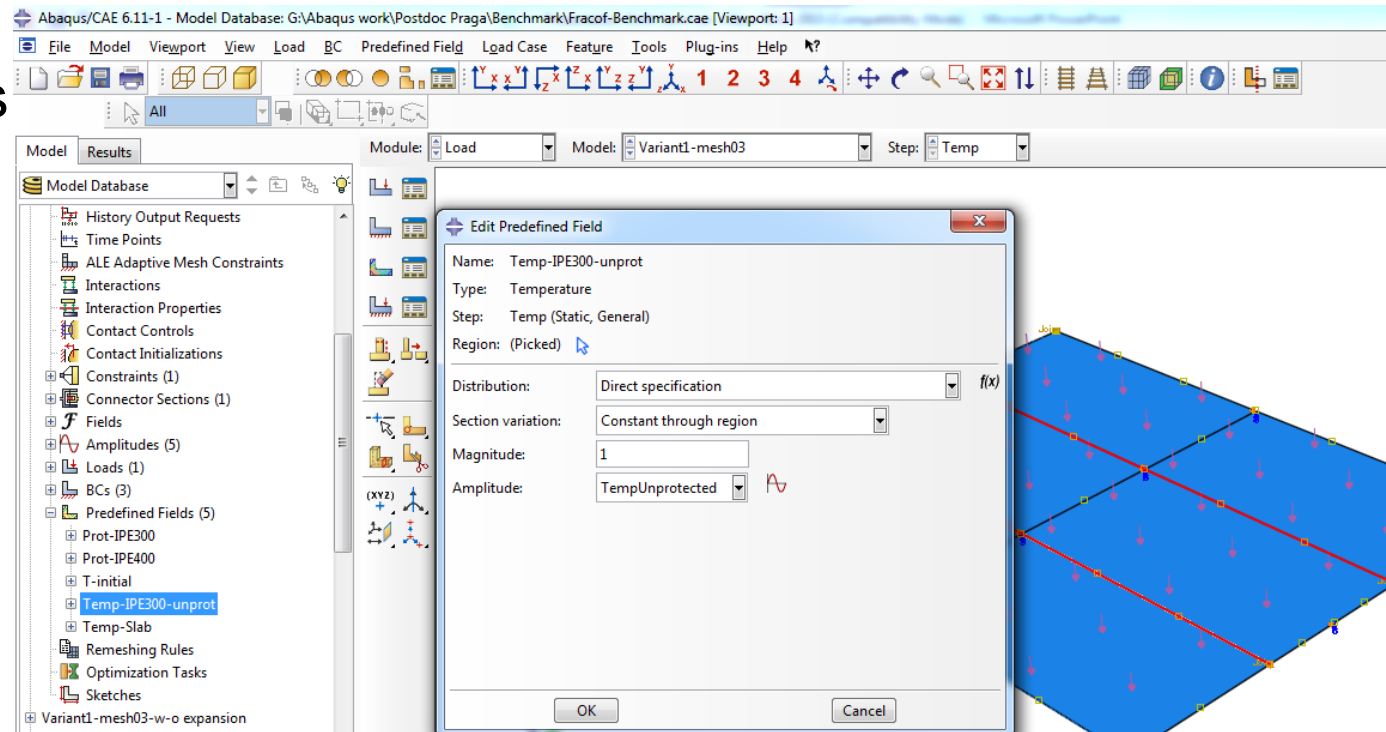
Predefined fields:

-Initial temperatures
(entire structure)

-Gradients through
beam section
(protected beams)

-Constant through
region (unprotected
beam)

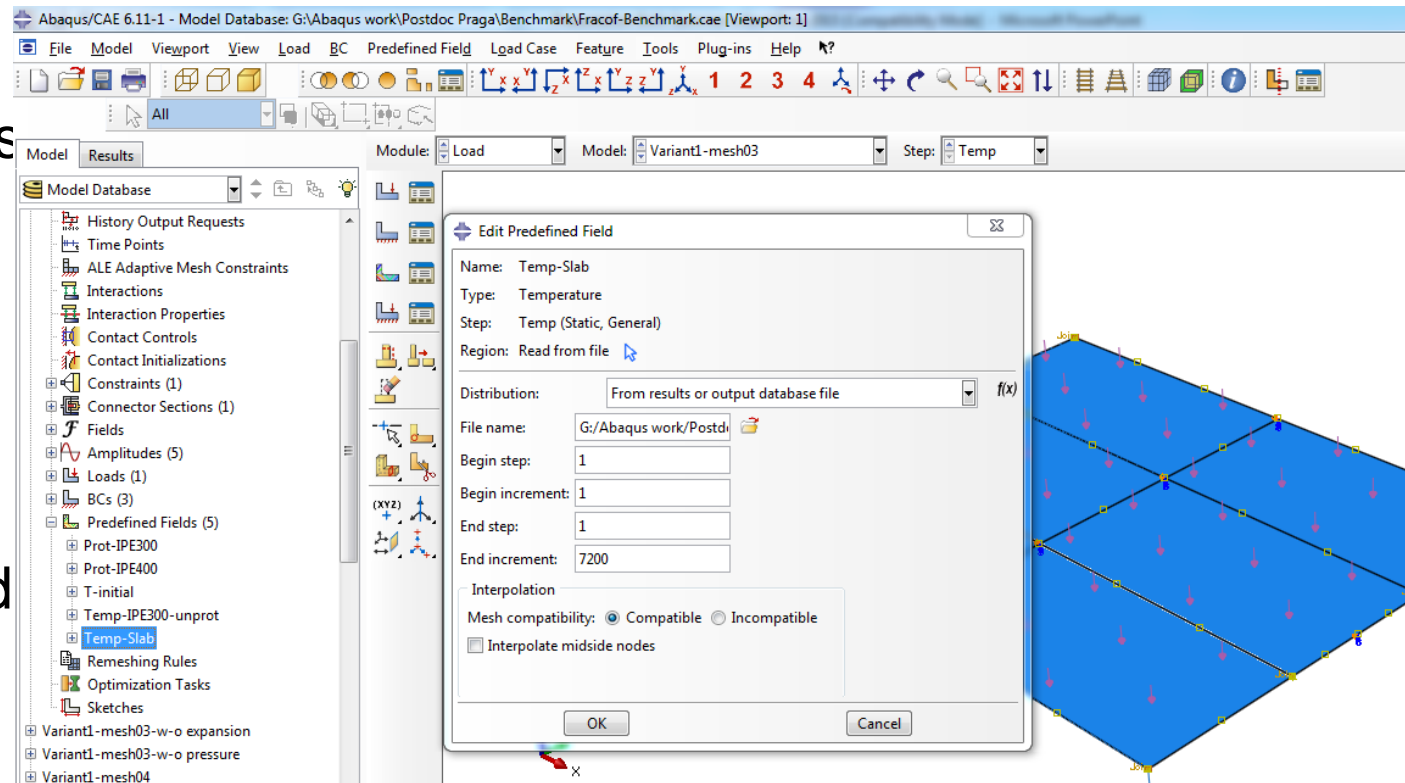
-From thermal
analysis data base
(slab)



Numerical model

Structural analysis of composite slab

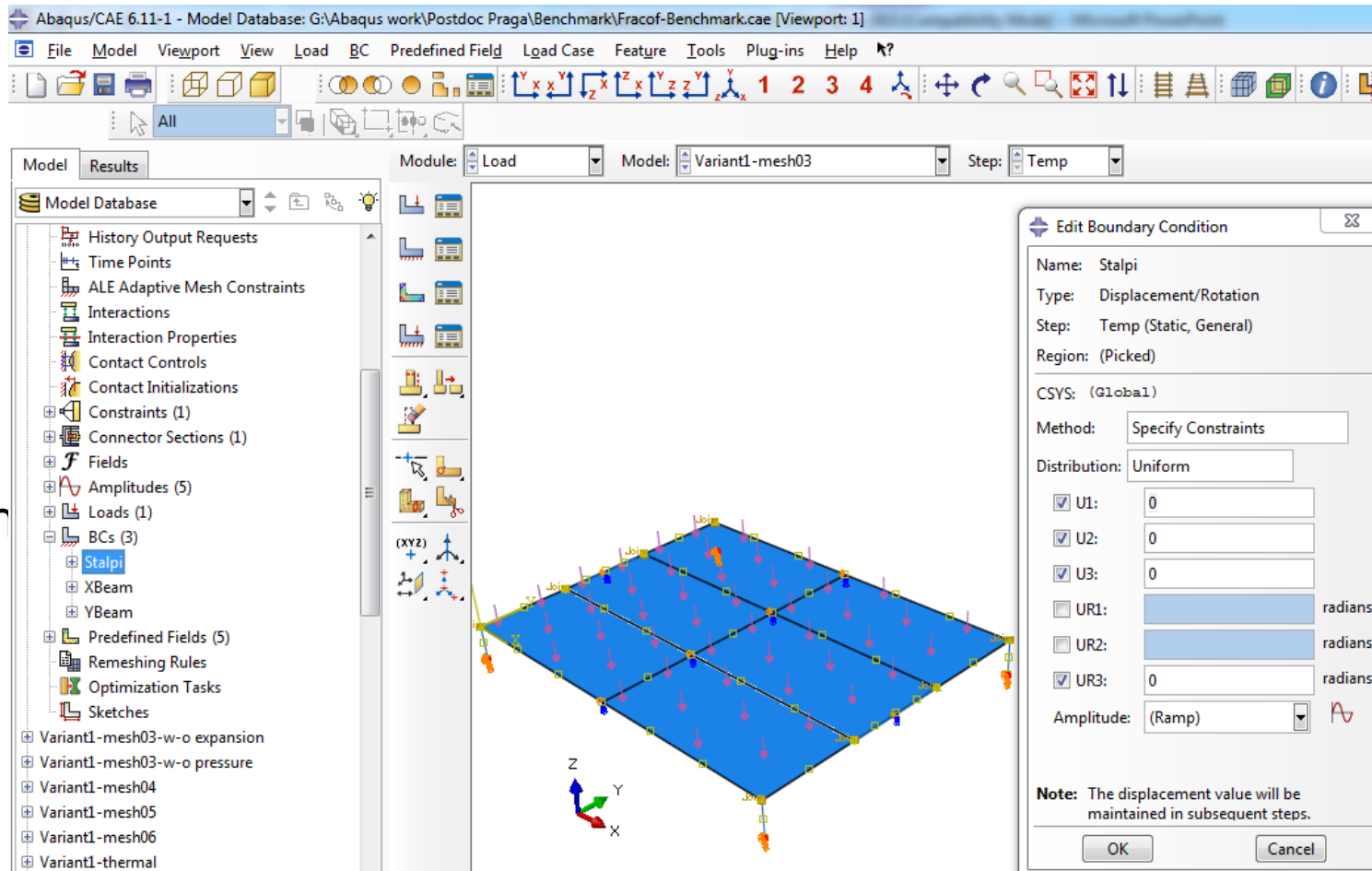
- Predefined fields:
- Initial temperatures (entire structure)
 - Gradients through beam section (protected beams)
 - Constant through region (unprotected beam)
 - From thermal analysis data base (slab)



Numerical model

Structural analysis of composite slab

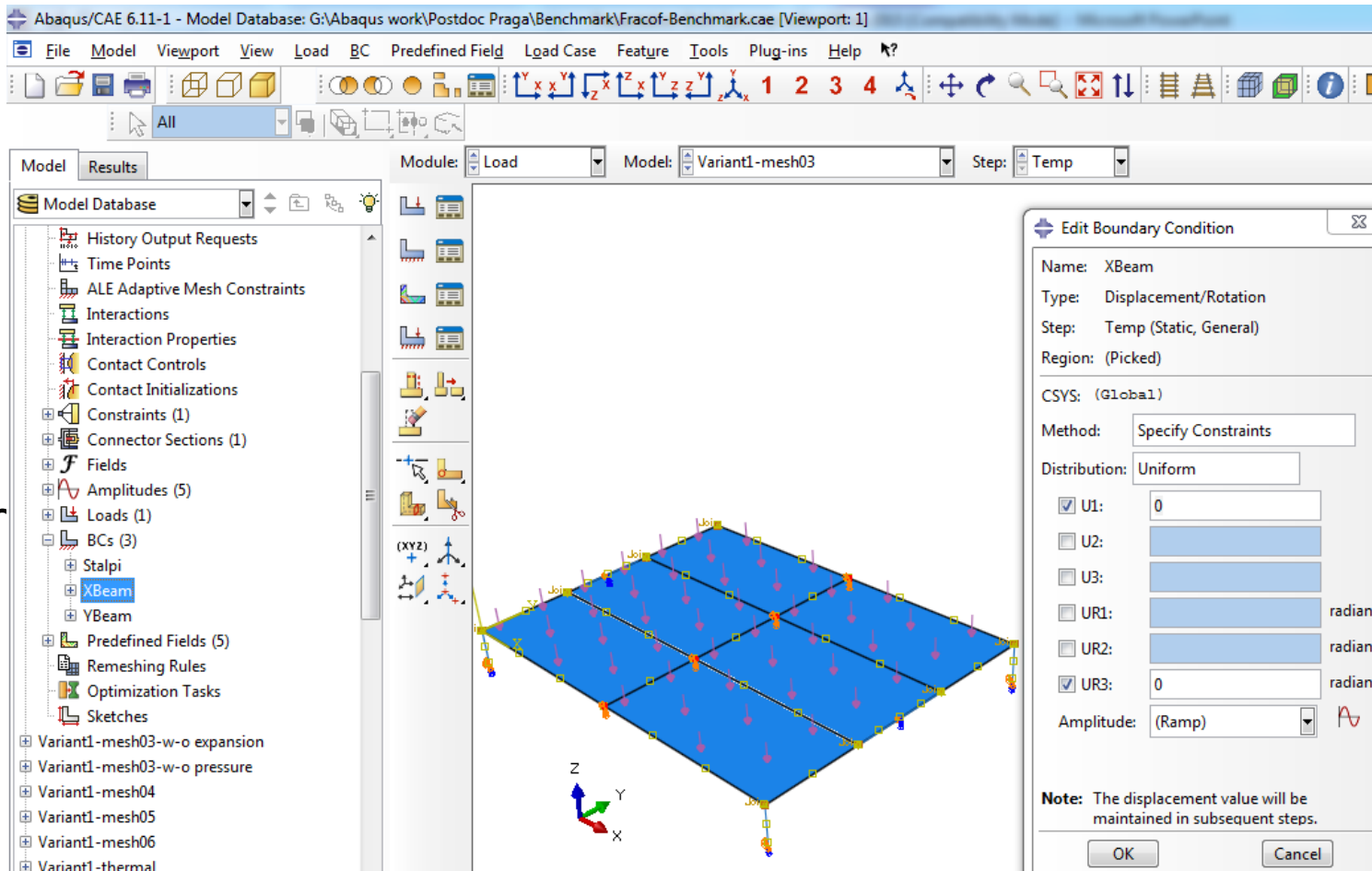
- Support conditions:
- Boundary conditions:
- columns
 - longit. direction
 - transv. direction



Numerical model

Structural analysis of composite slab

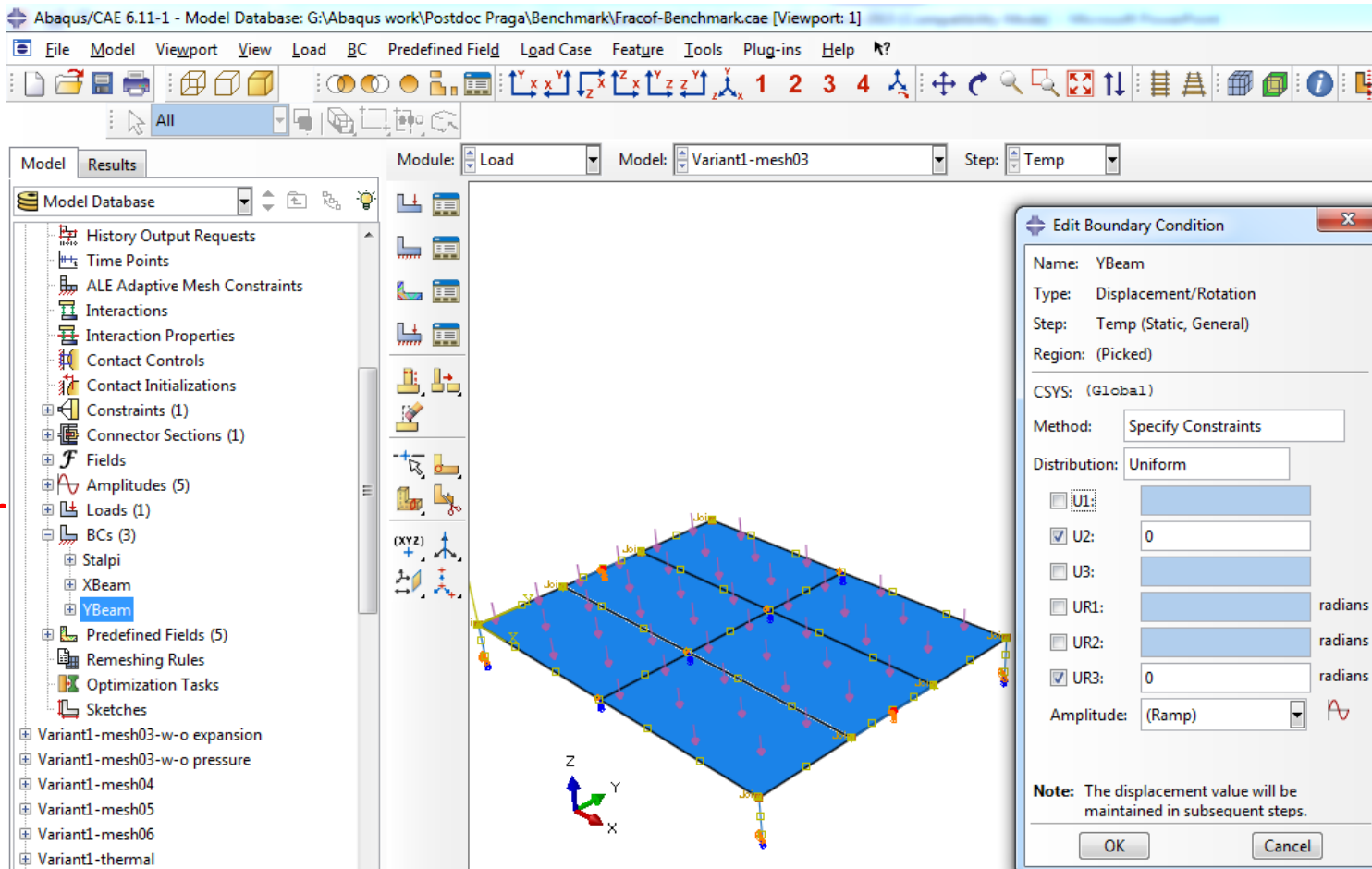
- Support conditions:
- Boundary conditions:
- columns
 - **longit. direction**
 - transv. direction



Numerical model

Structural analysis of composite slab

- Support conditions:
- Boundary conditions:
- columns
 - longit. direction
 - transv. direction

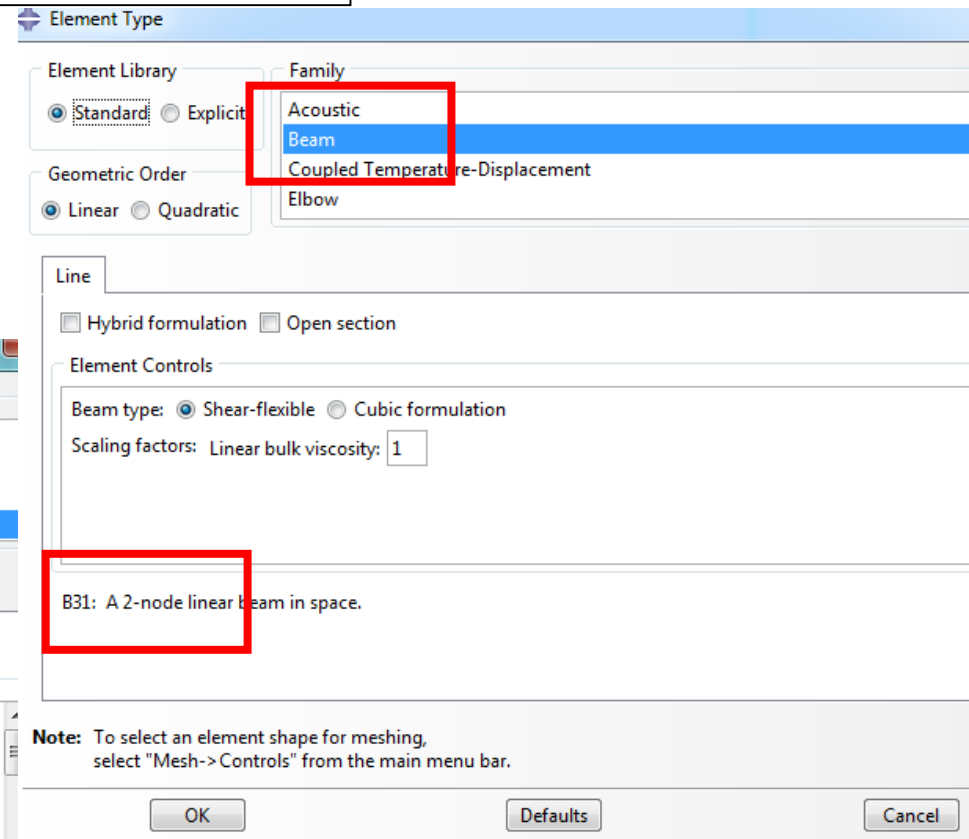
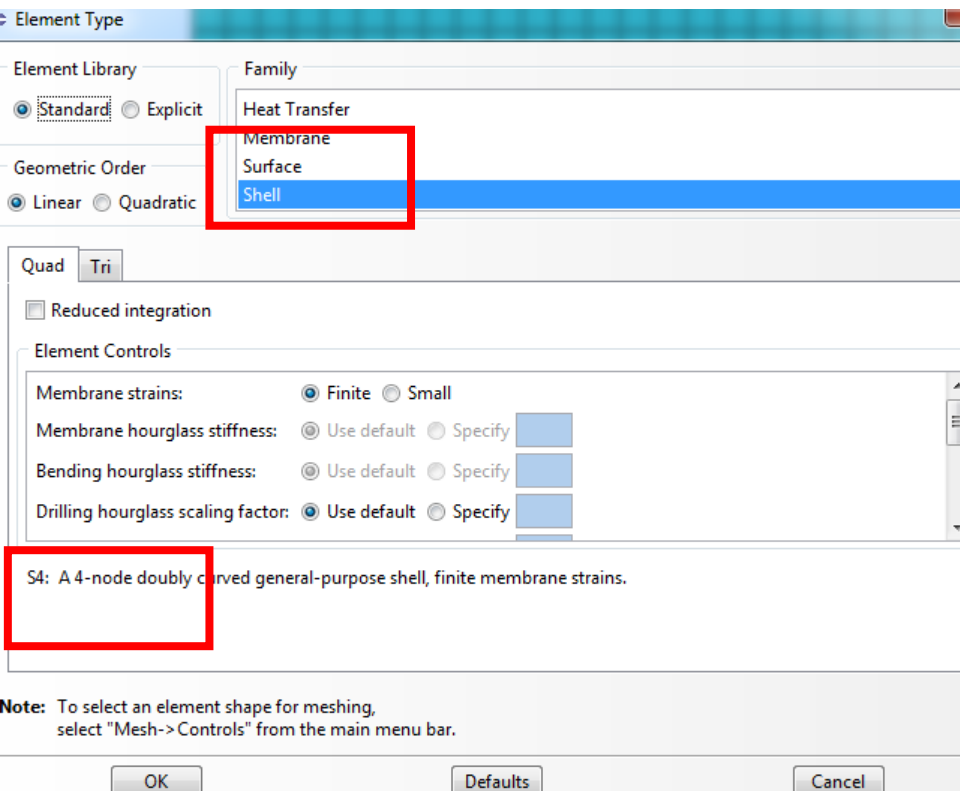


Numerical model

Structural analysis of composite slab

Mesh and finite elements:

Beams & →
columns: B31
(0.3 m)



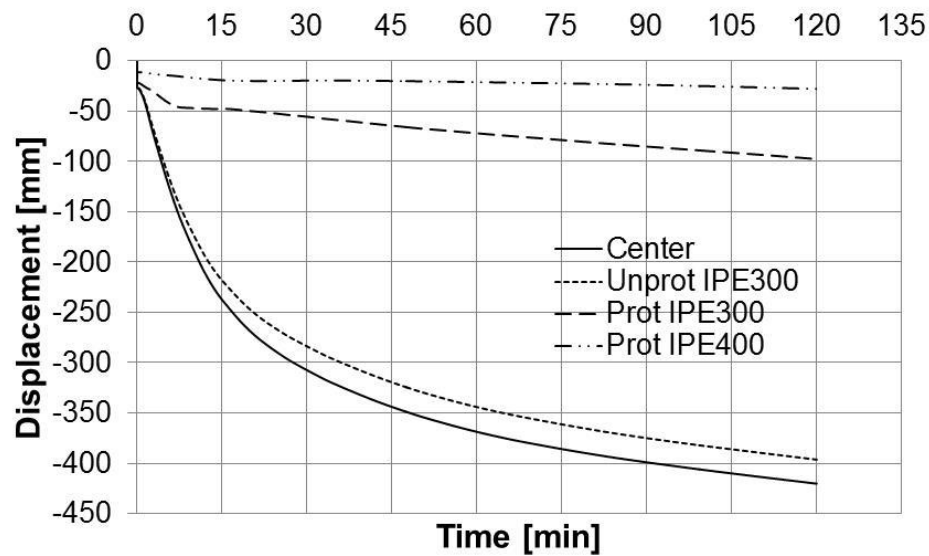
← Slab: S4
(0.3 m)

Numerical model - results

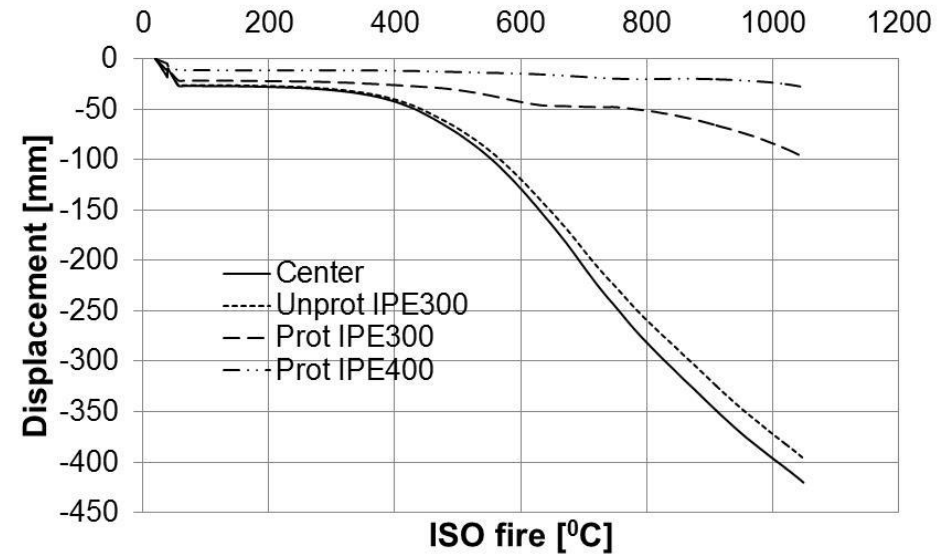
Structural analysis of composite slab

Results:

Displacements



With respect to time



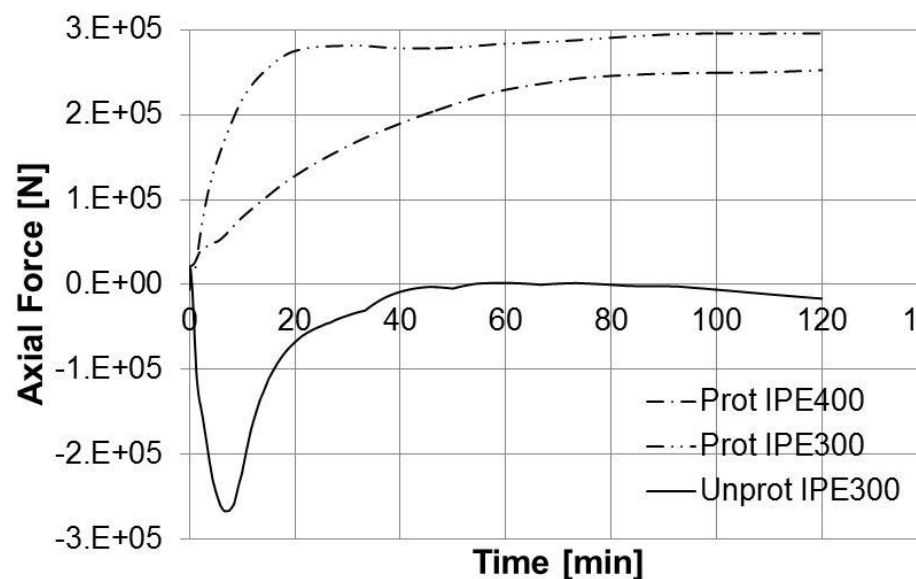
With respect to temperature

Numerical model - results

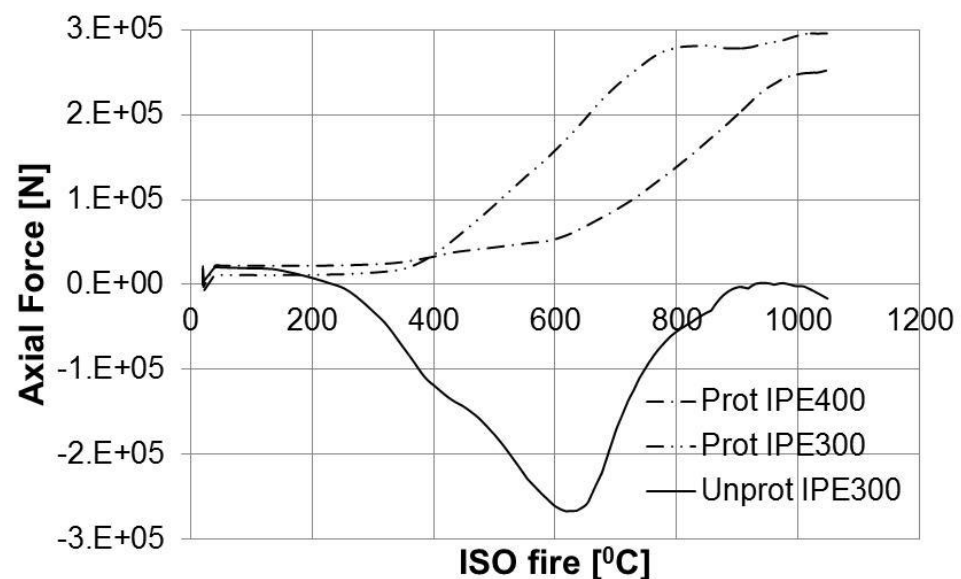
Structural analysis of composite slab

Results, other than obtained in experiment:

Axial force



With respect to time

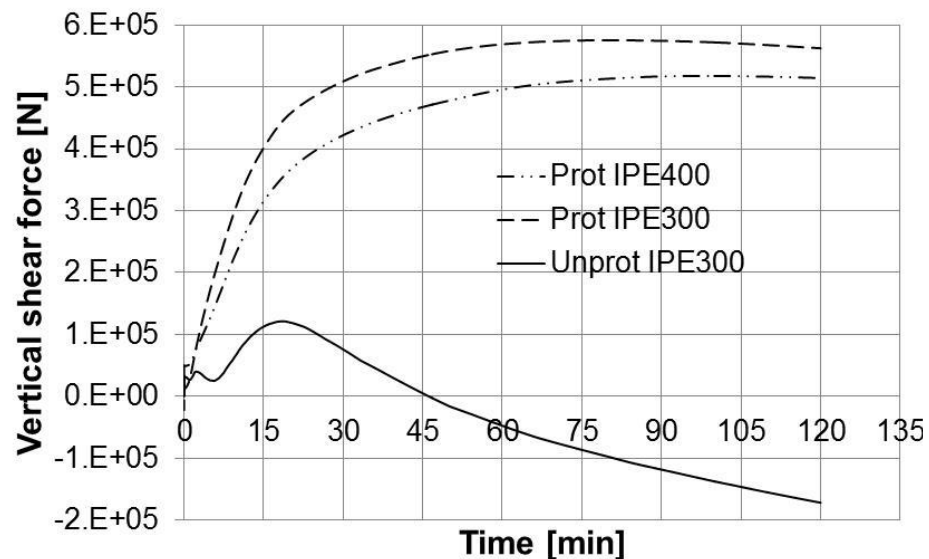


With respect to temperature

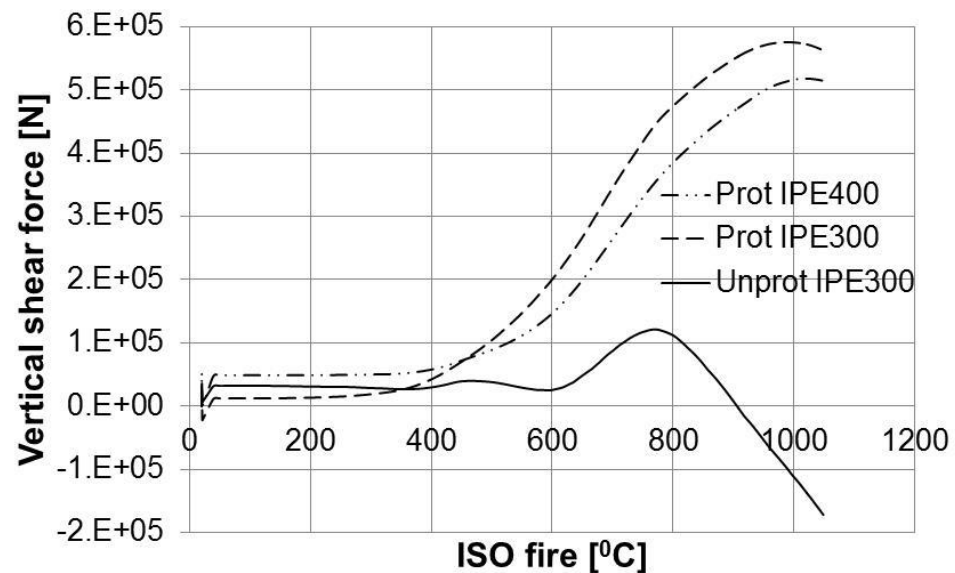
Numerical model - results

Structural analysis of composite slab

Results, other than obtained in experiment:
Vertical shear force



With respect to time

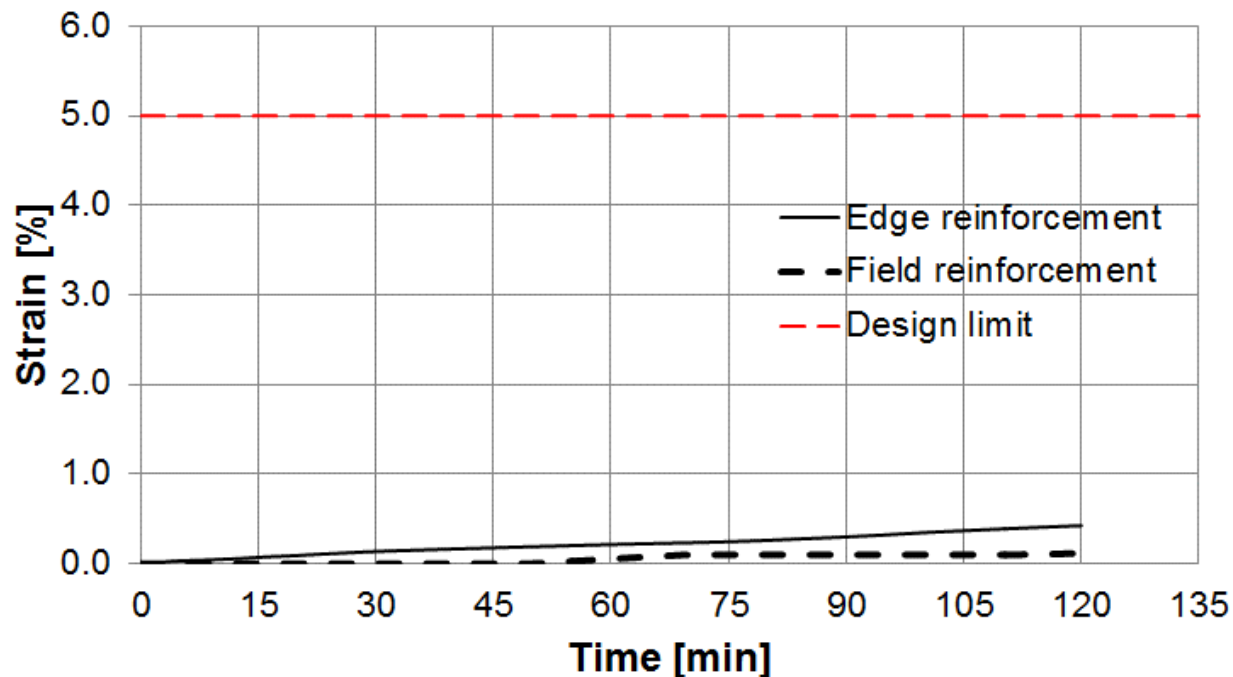


With respect to temperature

Numerical model - results

Structural analysis of composite slab

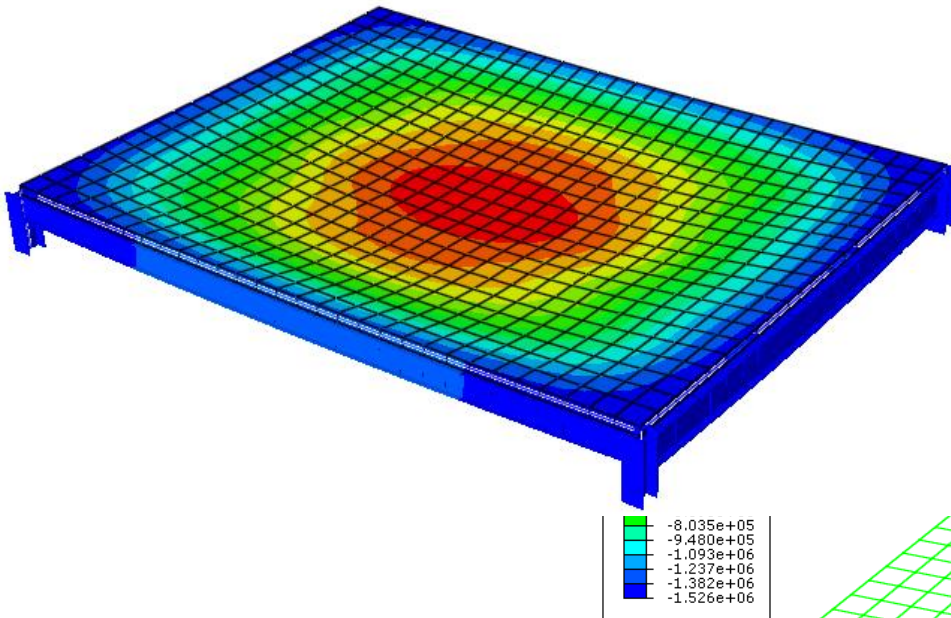
Results, other than obtained in experiment:
Reinforcement strain



Numerical model - results

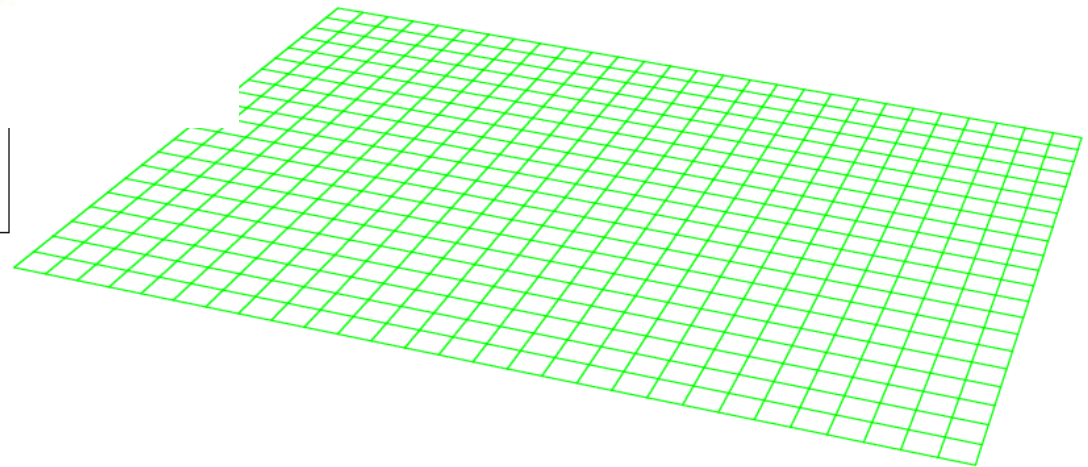
Structural analysis of composite slab

Deformed shape



Axial force

Scale Factor: +0.00



ODB: Variant1-mesh03.odb Abaqus/Standard 6.11-1 Wed May 27 19:02:15 GTB Daylight Time 2015

Step: Temp
Increment 1573; Step Time = 7200.
Symbol Var: SF
Deformed Var: U Deformation Scale Factor: +2.000e+00

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		

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

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

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

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

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

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

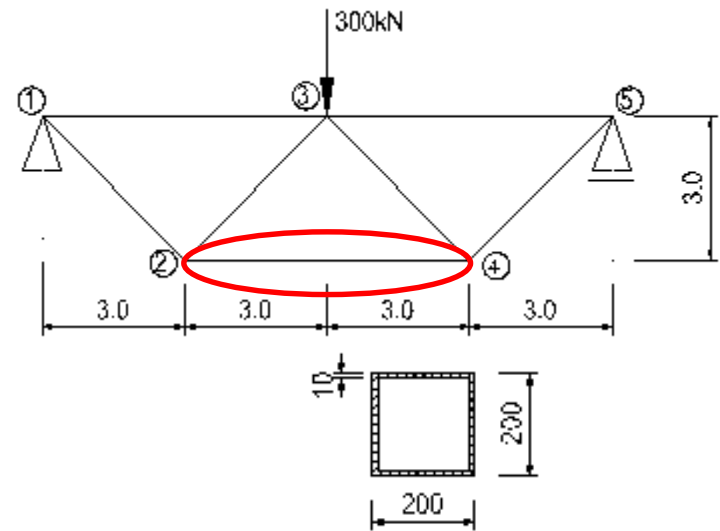
$$A_1 = \frac{f_1 - f_{h=0}}{f_{h=0}} \cong \frac{f_1 - f_2}{f_{h=0}} \frac{1}{r^p - 1}$$

- **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.**

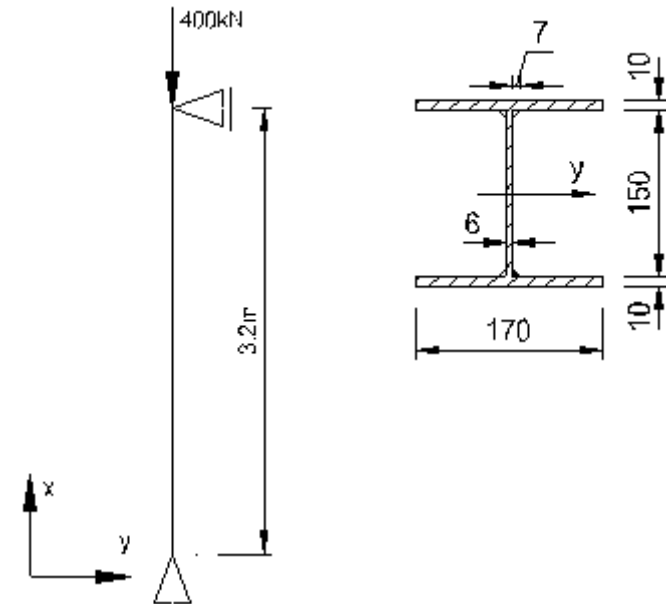
Bibliography

- [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

Tensioned element



Compressed element



Bended element

