FEM simulation of composite column during fire exposure

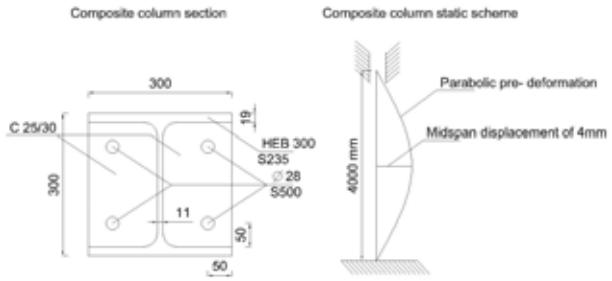
based on DIN EN 1991-1-2/NA:2010-03
using ANSYS software

TABLE OF CONTENTS

Software presentation

- CAE software based on FEM
- Initially designed for simulation of static, dynamic and thermal problems
- Founded 1970
- Nowadays consists of simulation tools: Structural Mechanics, Multiphysics, Fluid Dynamics, Explicit Dynamics, Electromagnetics, Hydrodynamics
- As well as of workflow: Workbench, Geometry Interfaces…

Assumpionts: Geometry description



Buckling coefficient – 0,5

Assumptionts: Reference Data

FAILURE TIME tu	[min]		92
HORIZONTAL MIDSPAN		30 min	4,4
DISLPLACEMENT AFTER TIME t		60 min	5,5

The temperature after fire time of 90 minutes:

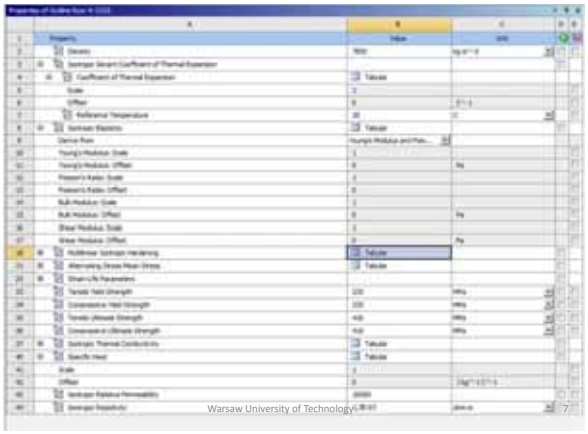
Reinforcement	535 C
Center of gravity of the steel profile	474 C

Assumpionts: Material parameters

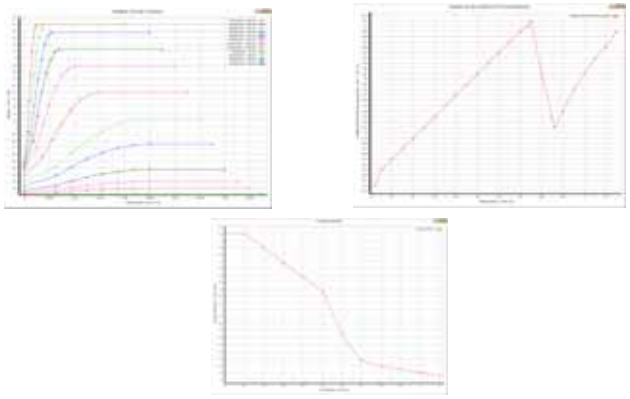
Concrete C25/30 (3 % moisture (by mass))		f_{ck}(20°C) N/mm²	25
Reinforcing steel S 500	fyk(20°C)	N/mm²	500
Structural steel S 235	fak(20°C)	N/mm²	235
Stress – strain curve		Concrete a. Reinforcing steel bars Structural steel	DIN EN 1994-1-2
Temperature load		ETK (four sides)	DIN EN 1991-1-2
Heat transfer coefficient	αc	W/(m² × K)	25
Emissivity	εm		0,7
Thermal and physical material properties		Concrete λ, ρ, c_p, $\varepsilon_{th,c}$ Steel λ, ρ_a, c_a, $\varepsilon_{th,s}$, $\varepsilon_{th,a}$	DIN EN 1994-1-2 DIN EN 1994-1-2

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

Assumpionts: Material parameters



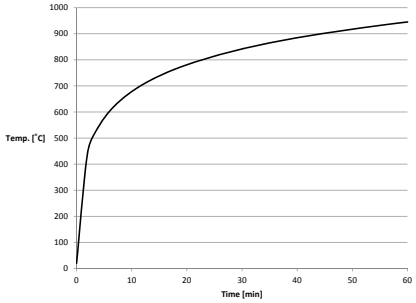
Assumpionts: Material parameters



Assumpionts: Load description

Standard ISO fire curve

T=20+345·log10(8t+1)



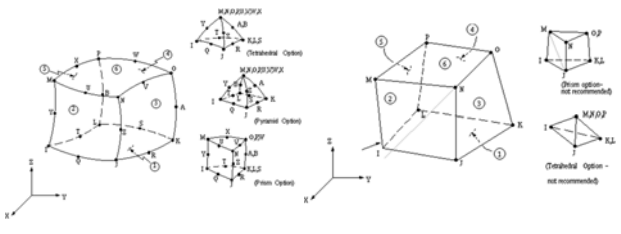
t	T
0	20
6	603,1176
12	705,4362
18	765,672
24	808,5173
30	841,7959
36	869,0098
42	892,0323
48	911,984
54	929,5883
60	945,3401
66	959,5922
72	972,6057
78	984,5786
84	995,6652
90	1005,988

Analysis: schematics

- Same geometry
- Same mesh
- Same engineering data
- Different finite elements
- Load transfer

Analysis : mesh and elements

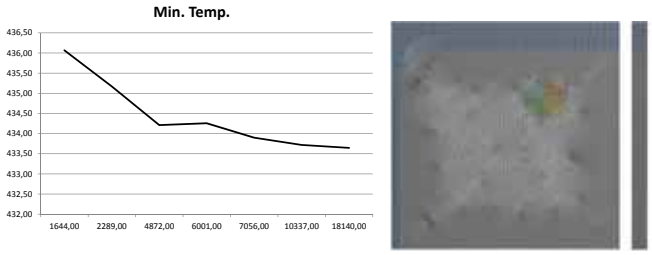
SOLID 90 SOLID 185



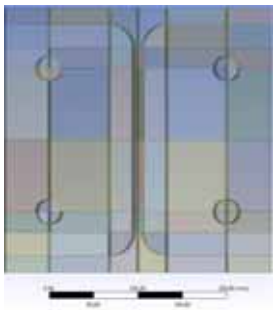
Source: ANSYS MANUAL

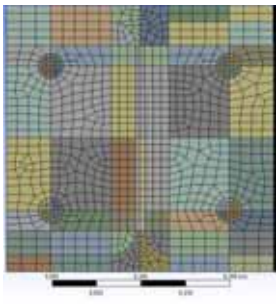
Analysis : mesh and elements

Analysis nr	Element Type	Element size [m]	Number of Elements	Min. Temp.
1	shell 57	0,02	1644,00	436,07
2	shell 57	0,01	2289,00	435,17
3	shell 57	0,007	4872,00	434,21
4	shell 57	0,006	6001,00	434,26
5	shell 57	0,005	7056,00	433,90
6	shell 57	0,004	10337,00	433,72
7	shell 57	0,003	18140,00	433,65



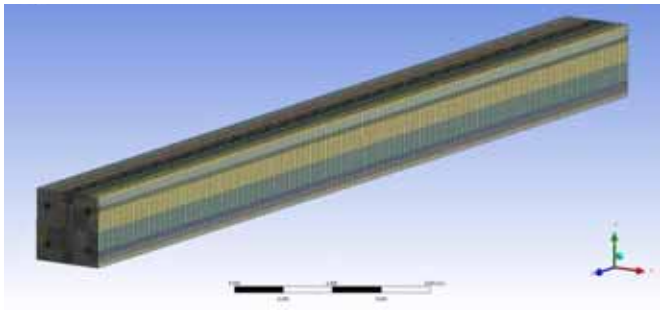
Analysis : mesh and elements



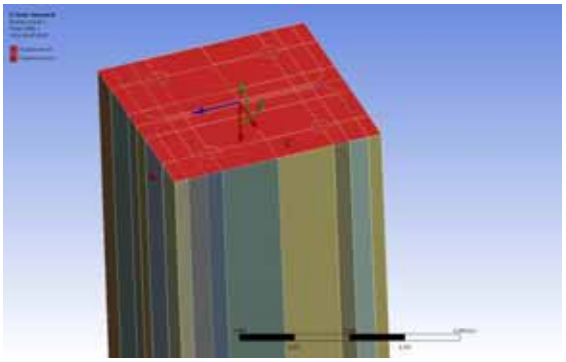


184 440 finite elements on column

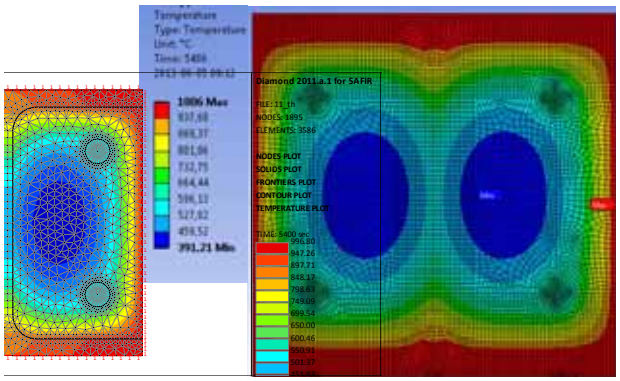
Analysis : mesh and elements



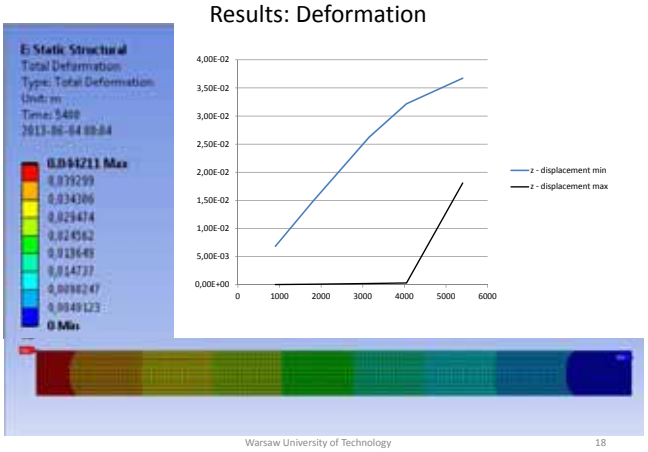
Analysis: boundary conditions



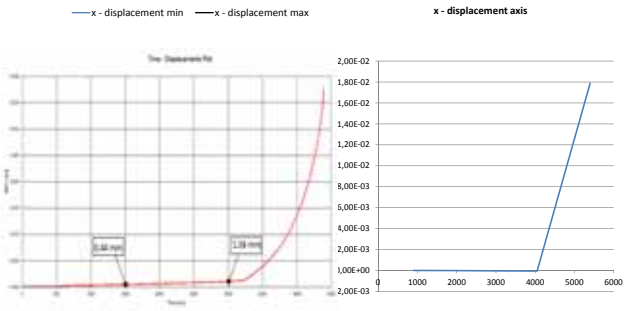
Results: Temperature



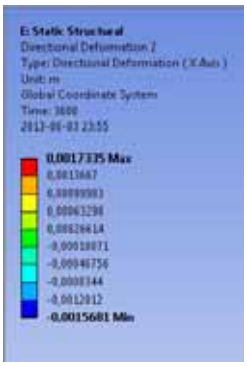
Results: Deformation

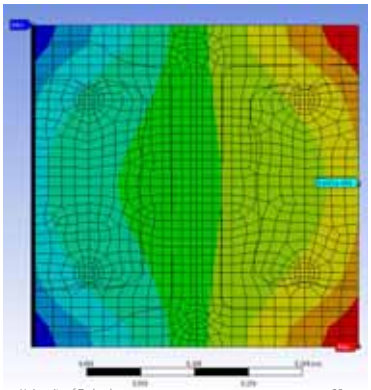


Results: Deformation



Results: Deformation





Conclusions

- mesh sizing is secondary factor when it comes to temperature field

- there is no difference in temperature result in 2D and 3D analysis beside computation cost

- the problem of applying temperature field on 2D model of cross- section on the model for static analysis

- differences in temperature mainly affected by the load application

TIPS FOR THE FUTURE:

- Find a way to apply 2D results of heat transfer analysis for the 3D model in ANSYS software to save computational cost
- Create standard schematic workflow in Ansys Workbench to conduct range of analysis for different geometry or engineering data