Validation of CFD Model of Spent Fuel Pool at NPP Temelin

Autors: Ondřej Burian, Pavel Zácha, Václav Železný

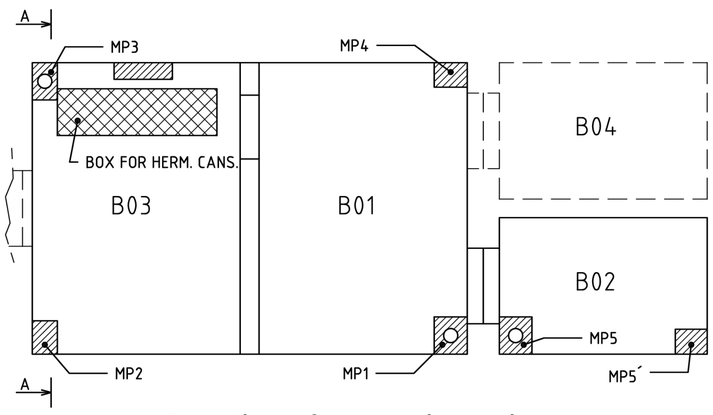
Czech Technical University in Prague, Faculty of Mechanical Engineering, Dpt. of Energy Engineering

Introduction

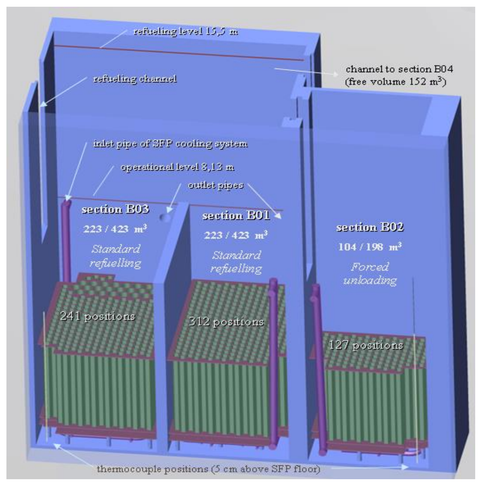
- CFD computational model of spent fuel pool (SFP) of NPP Temelín ⇒ Created for analysis of temperature distribution in spent fuel pool in normal and abnormal operation conditions by company TES s.r.o in cooperation with FME CTU in Prague
- Summer 2011 – measurement of vertical temperatures profiles during shutdown at 1.Unit NPP Temelín by TES s.r.o
- Spring 2012 – validation of CFD model based on measured data by CTU in Prague

Spent fuel pool description

- Overall volume 1050 m^3 (14,5 x 6 x 15,5 m)
- Storage spent fuel in compact storage grid
- Division to three sections – B01, B02, B03



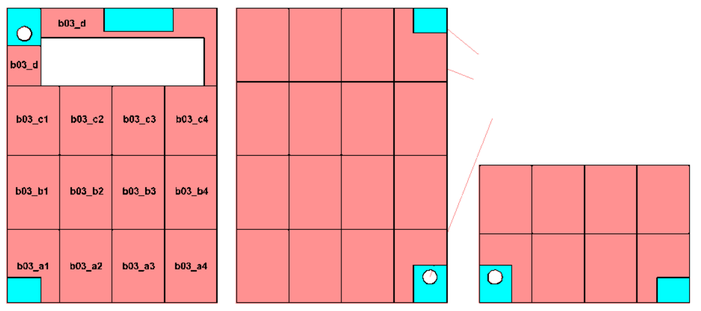
Location of measuring points during validation measurement

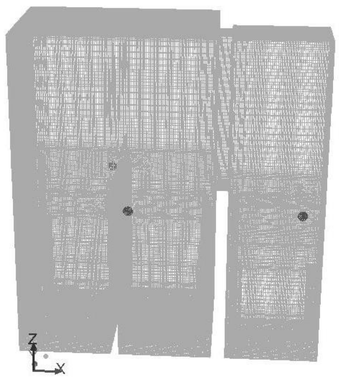


Disposition of NPP Temelin spent fuel pool

Description of CFD model of spent fuel pool

- Number of cells limited to 1 milion – computational time
- Large volumes of cells – average length of edges 100 mm
- Application mostly structured grid
- Heat transfer by walls not considered – adiabatic wall
- Heat transfer by water level – 0,9 kW.m^{-2}
- In upper part set region with laminar flow
- Significant simplification against reality
- Storage grids modeled like porous zone

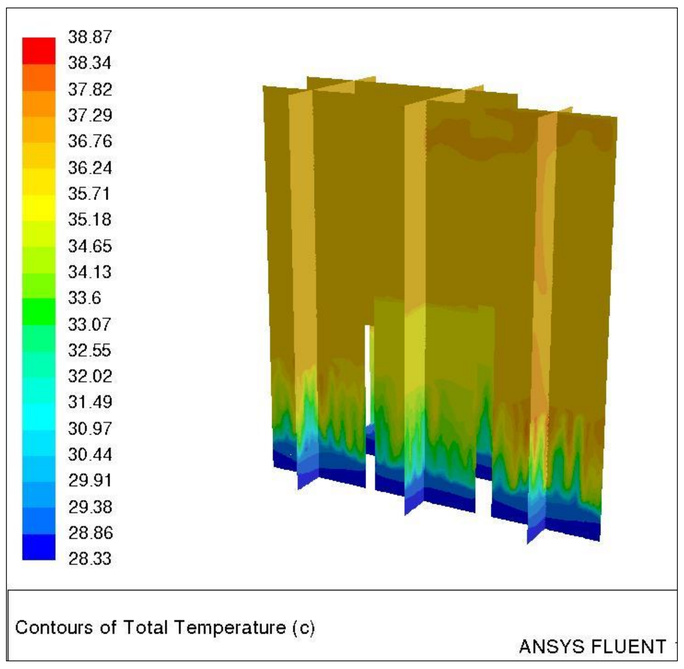




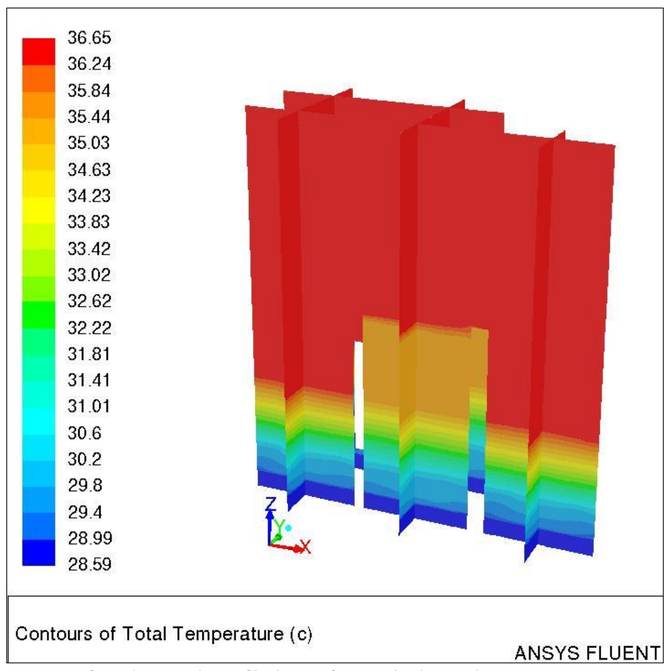
CFD model nodalization

Validation calculation

- Problem wit divergence of calculation ⇒ calculation in stages with different set of solver and settings laminar flow in upper region



Results from segregated solver („pressure based“) – pre calculation



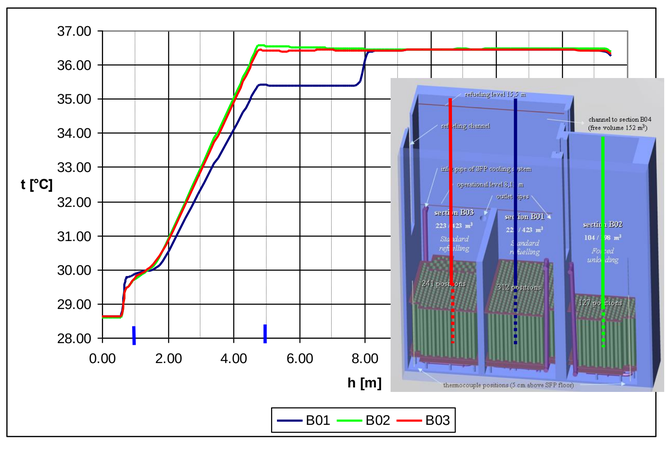
Results from implicit solver („density based“) – final calculation

Final computation strategy

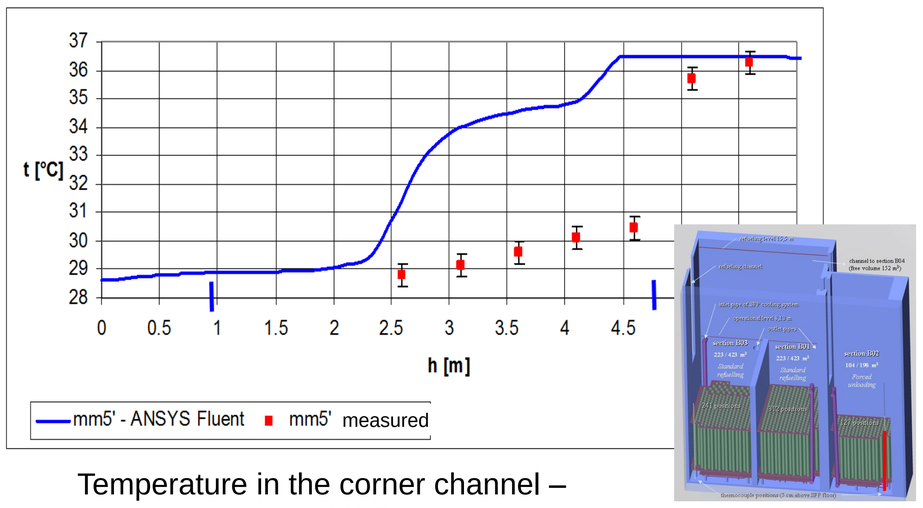
iteration interval	accuracy of computation	turbulence solver	„courant number“
0 – 5 000	**normal**	**„pressure based“**	**-**
5 000 – 50 000	**„double precision“**	**„pressure based“**	**-**
50 000 – 100 000	**„double precision“**	**„density based“**	**0,5**
100 000 – 130 000	**„double precision“**	**„density based“**	**2**

Computation was provide on 8 quad-core processors AMD Opteron 8354 – 2,2 GHz. Time of computation 1040 hours machine time

Results of validation computation of CFD model



Temperature in the center of pool sections



Temperature in the corner channel – measuring point mMP5‘ – measuring point with worst accordance model and measured data

Conclusion to existing CFD model:

- In area of corner channel is model described insufficiently.
- In area of storage grid is assume good accordance model with reality.
- In area up the storage grid was confirm good accordance model wit reality

Conclusion to improve the CFD model:

- Better settings heat transfer between corner channel and storage grid
- Refining the CFD grid in corner channel area
- Searching a use better model of turbulences which better describe flow conditions in spent fuel pool

Acknowledgment

This work was supported by the Grant Agency of the Czech Technical University in Prague, grant No. SGS13/133/OHK2/2T/12

Contact

- Ing. Pavel Zácha, Ph.D.
- Ing. Martin Blaha (TES s.r.o)

ondrej.burian@fs.cvut.cz