CFD fire simulations

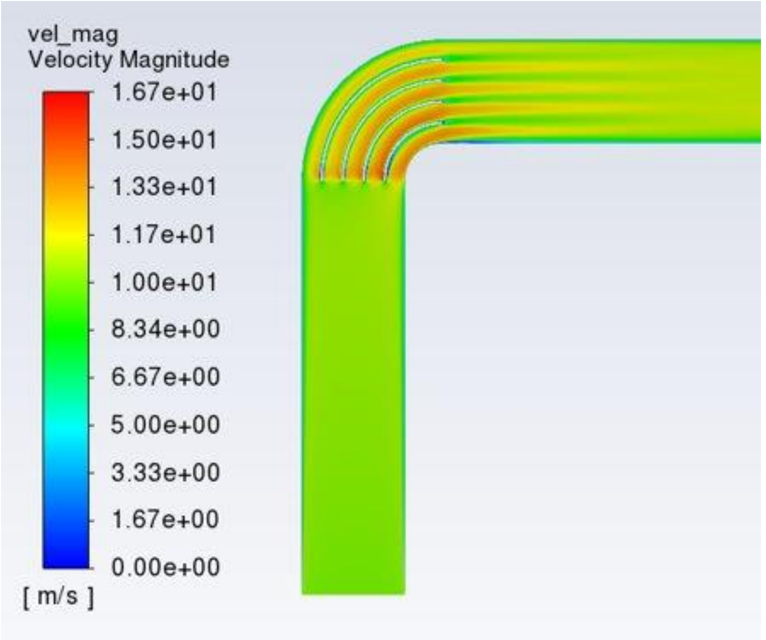
UNIVERSITY OF CHEMISTRY AND TECHNOLOGY PRAGUE

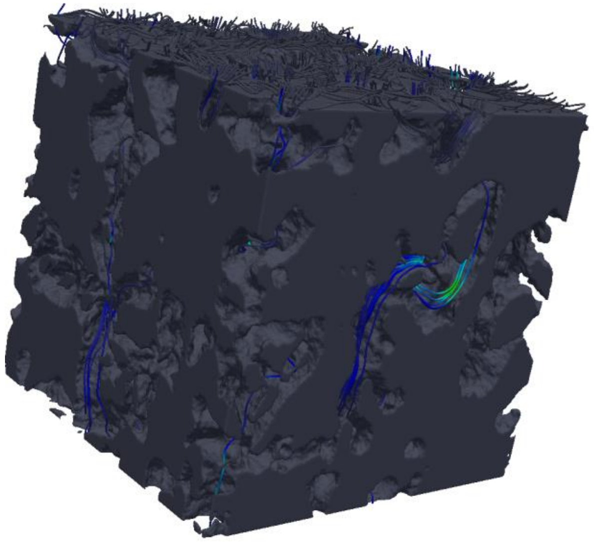
Aleš Palkovič

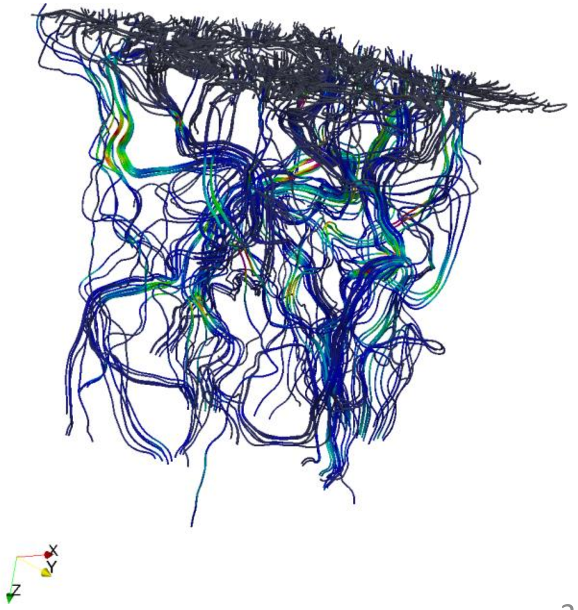
Labπ

What is Computational Fluid Dynamics?

- Enhanced method for solving fluid flow problems
- Numerical solution of Navier-Stokes equations
- Other models for heat transfer, turbulence, reactions and more

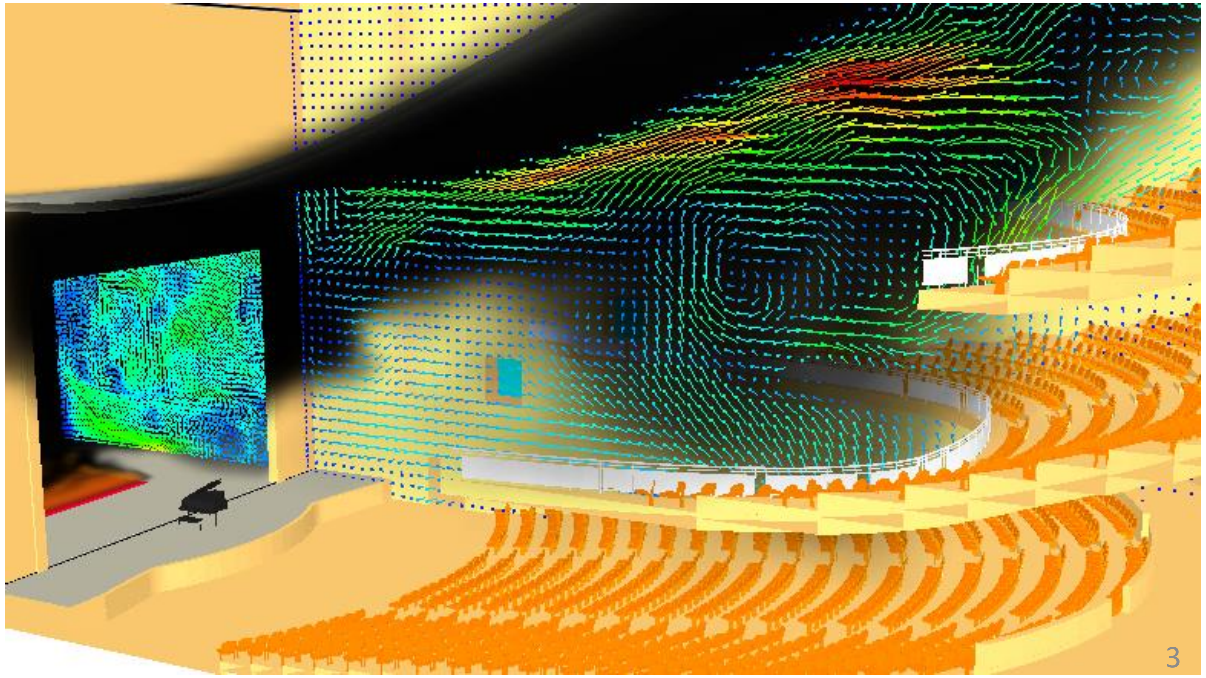






CFD in fire engineering - *extension of fire tests*

- Provides information about whole domain
- Different configurations of fire scenarios
- Non-destructive method



https://ignis.solutions/services/cfd-fire-smoke-modelling/

Performance based design

- Requirements on constructions and fire protection systems

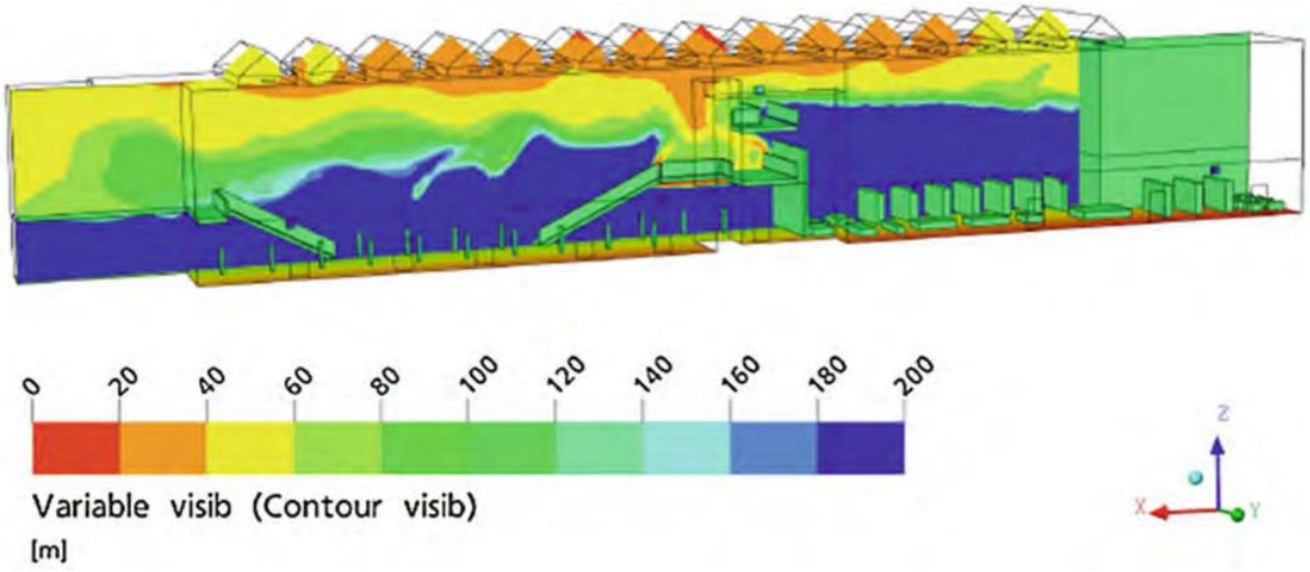


Fig. 32.5 Map of visibility in a vertical plane at time = 3 min for a fire in a library building (Figure courtesy: ANSYS Europe Ltd)

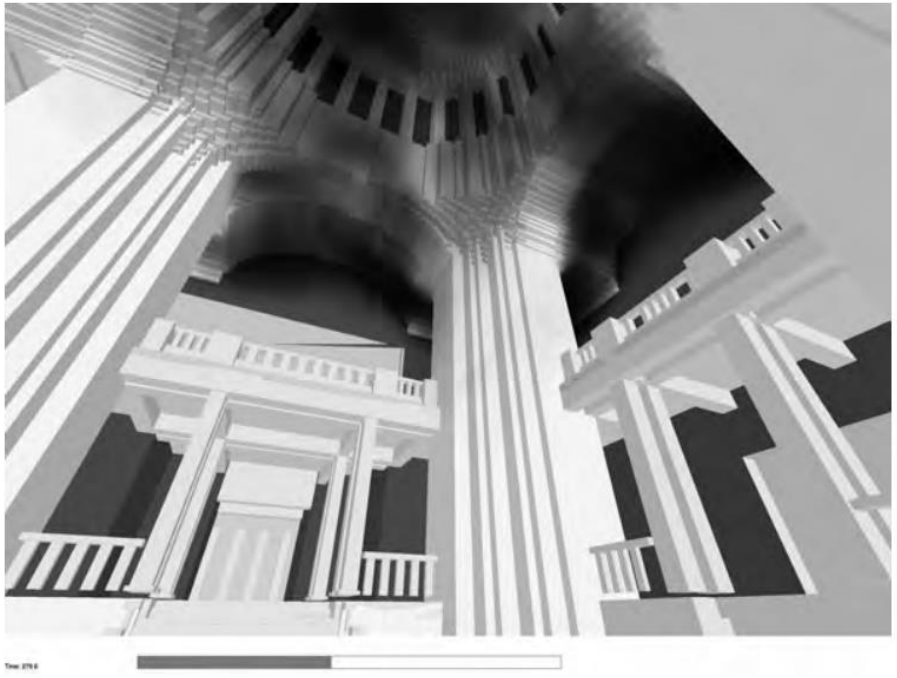


Fig. 32.6 Smoke filling analysis of the Rhode island State capitol (Figure courtesy: Hughes Associates)

Images from SFPE Hanbook [1]

Fire investigation

- WTC – NIST investigation report [2]

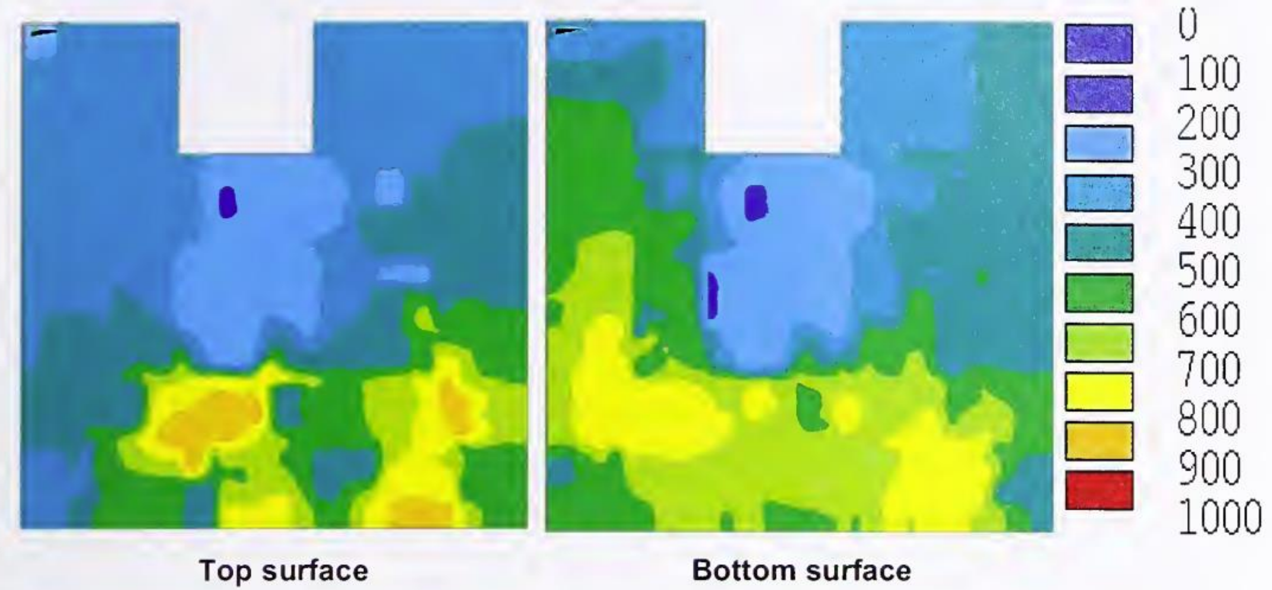


Figure 2–11. Temperature contours (°C) on the top and bottom faces of the concrete slab (96th floor, WTC 1) at 100 min after impact. A portion of the concrete slab on the north face (top) was damaged by the impact of the aircraft.

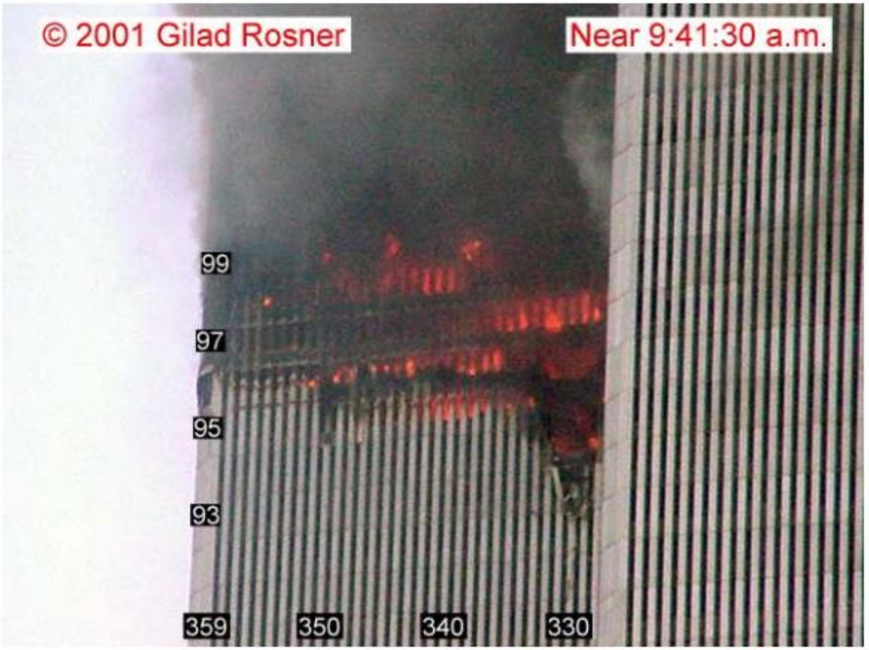


Figure 3–15. Photograph of the south face of WTC 1 taken at approximately 9:41:30 a.m. on September 11, 2001. Column and floor numbers were added to the original photograph.

Fire modelling - *a complex problem*

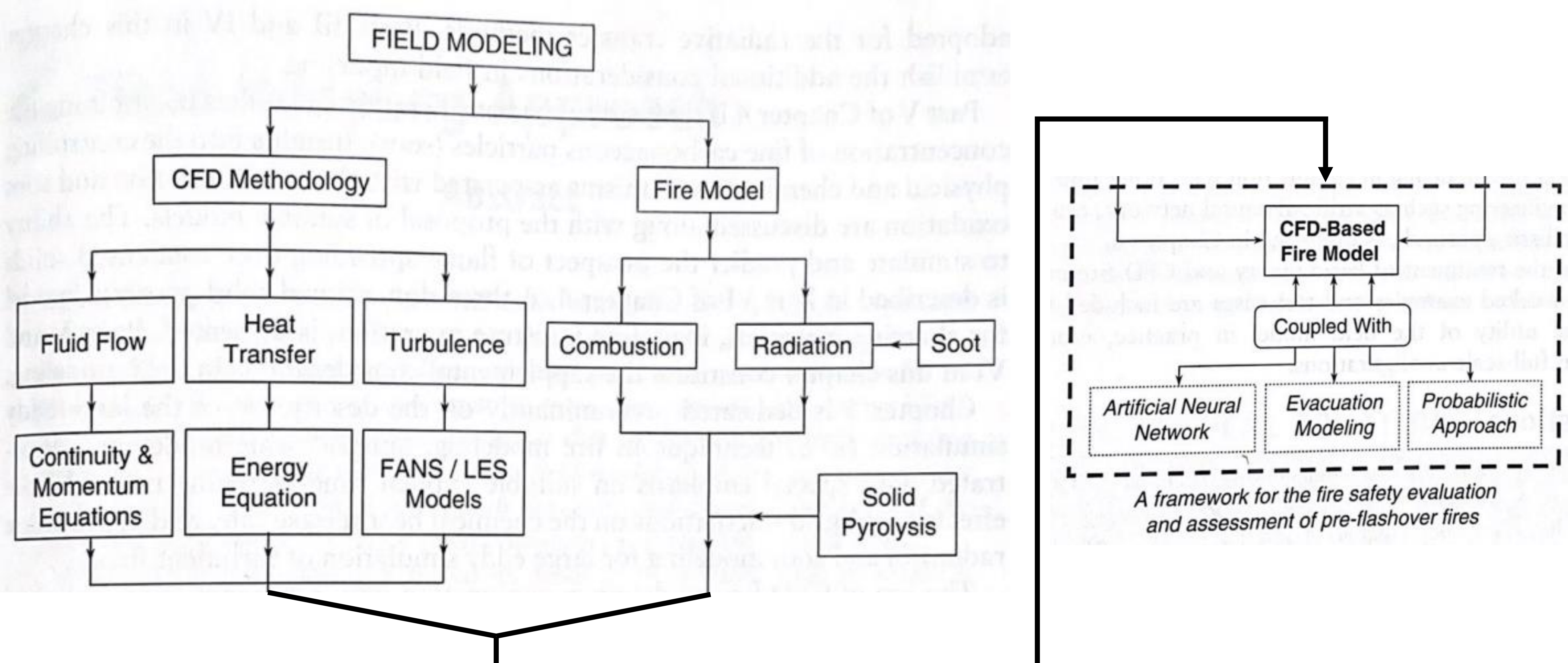


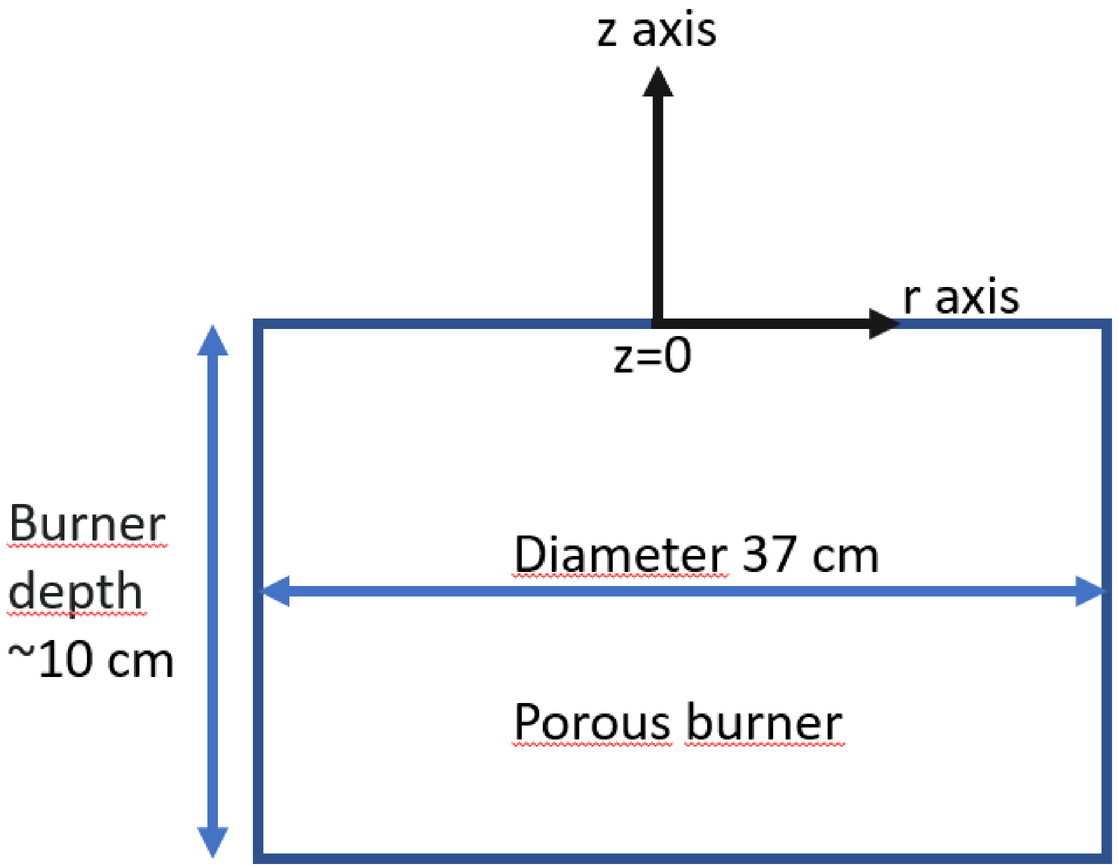
Image from [3]

MaCFP working group

- Fire research community initiative under International Association for Fire Safety Science (IAFSS)
- Effort to make systematic progress in fire modelling
- Based on well described experiments
- Categories: Gaseous pool fires, Liquid pool fires, Buoyant plumes, Wall fires, Extinction
- Participants: NIST, FM Global, universities and others
- Softwares: FDS, OpenFOAM or in-house codes

NIST experiment

- Gaseous pool fire
- Methane burner
- Defined flowrate
- Measured along z:
  - Temperature
  - Velocity
  - Species concentration



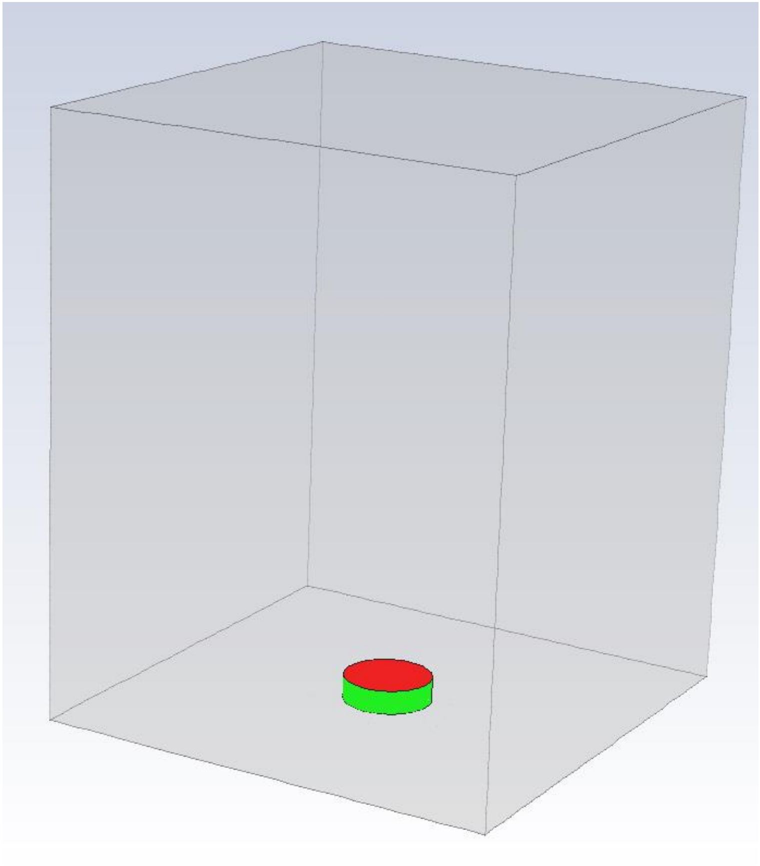
CFD model – Ansys Fluent

- Commercial software
- No solution in MaCFP group
- Universal – not primarily for fire engineering problems
    - <u>Great variety of models</u>

- My goal: recognize appropriate models and setting for fire engineering purposes

Domain

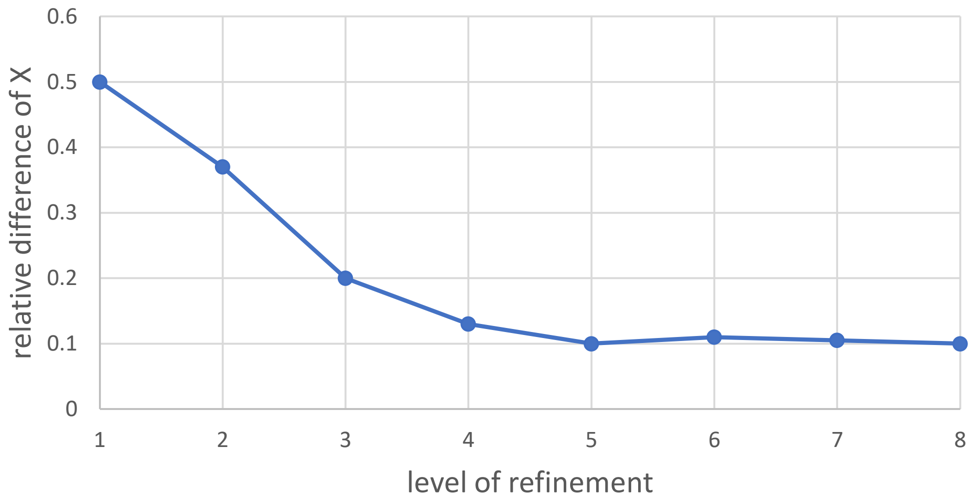
- Inlet (red)
- Wall (green)
- Pressure outlet (grey)



- What are the perfect dimensions of the domain?

Mesh size

- For the FVM
- Balance between accuracy and computation time



- Refining crutial regions of domain
  - How many cells should be at the fuel inlet?
  - How much do we refine the mesh around the flame?

Turbulence model

RANS

- Averaged fluctuations
- Used in industry
- Computationally cheap

LES

- Large eddies directly computed
- Used in fire engineering
- Computationally expensive
- Detailed

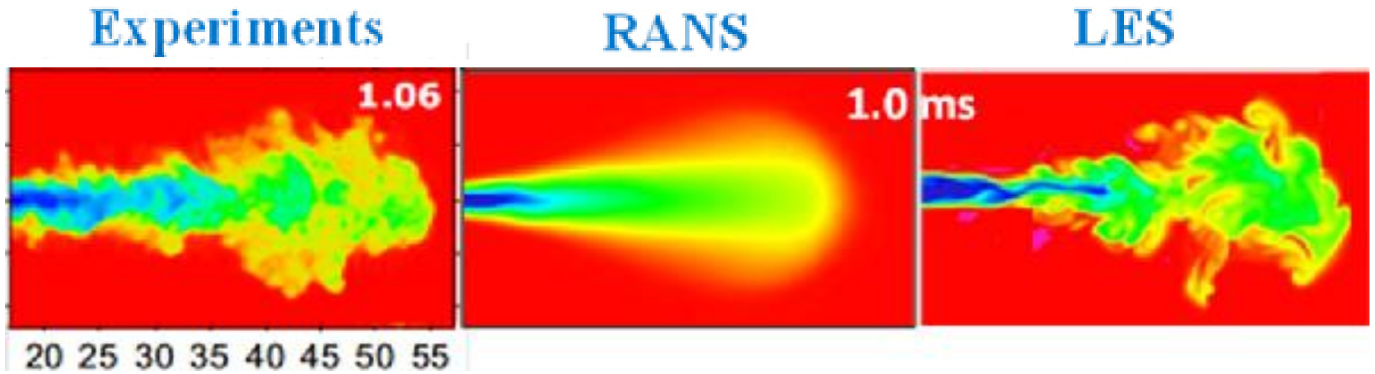


Image from [4]

Steady state or transient simulation?

$$\underbrace{\frac{\partial(\rho\phi)}{\partial t}}_{\text{time derivative}} + \underbrace{\nabla\cdot(\rho\boldsymbol{u}\phi)}_{\text{convection term}} = \underbrace{\nabla\cdot(\Gamma\nabla\phi)}_{\text{diffusion term}} + \underbrace{S_\phi}_{\text{source term}},$$

Steady state

- Simplification
- Cheap to compute
- Only for specific problems

Transient

- Process modeled in time
- Expensive (small time steps)
- It can reach the steady state

Can we consider the fire engineering problems to be steady state?

Combustion model

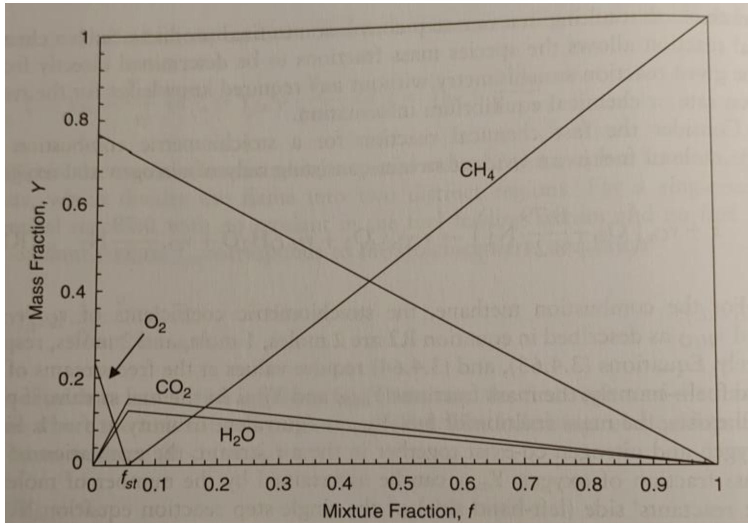
- Great variety in Fluent
- <u>Finite rate model</u>
  - Detailed mechanism, Arrhenius equation, expensive

$$\hat{R}_{i,r}=\Gamma\left(\nu''_{i,r}-\nu'_{i,r}\right)\left(k_{f,r}\prod_{j=1}^{N}\left[C_{j,r}\right]^{\eta'_{j,r}}-k_{b,r}\prod_{j=1}^{N}\left[C_{j,r}\right]^{\eta''_{j,r}}\right)$$

- <u>Eddy dissipation model</u>
  - Reaction rate based on mixing, two step reaction at max.

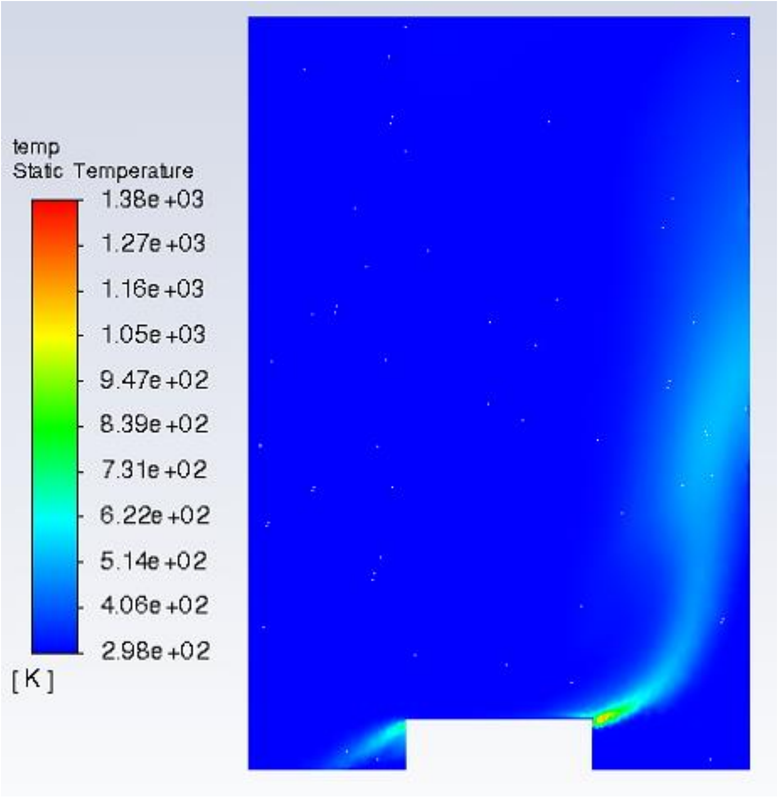
$$\overline{R}_{fu}=C_R\overline{\rho}\,\frac{\varepsilon}{k}\left(\widetilde{Y''^2_{fu}}\right)^{1/2}$$

- <u>Conserved scalar model</u>
  - Species represented by mixture fraction
  - immediate reaction
  - fluctuations handled via probabilistic approach (PDF tables)



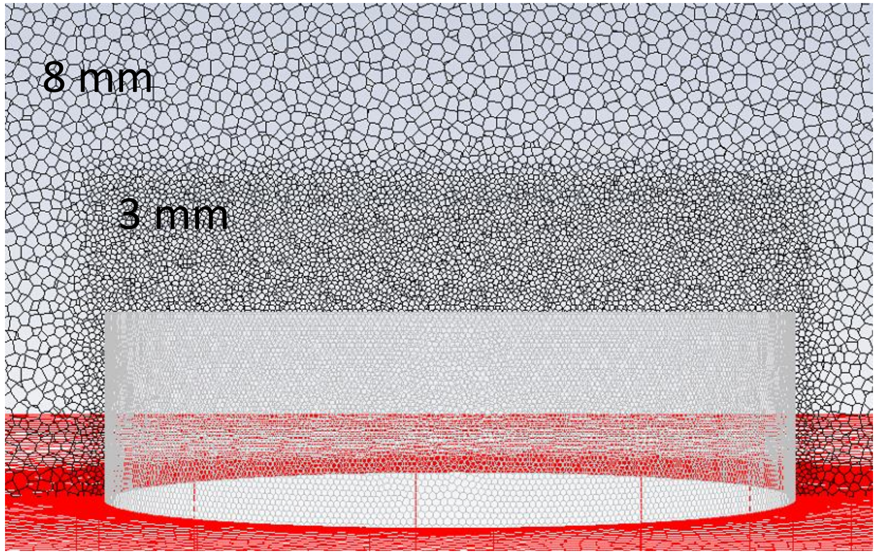
Results – First Attempts

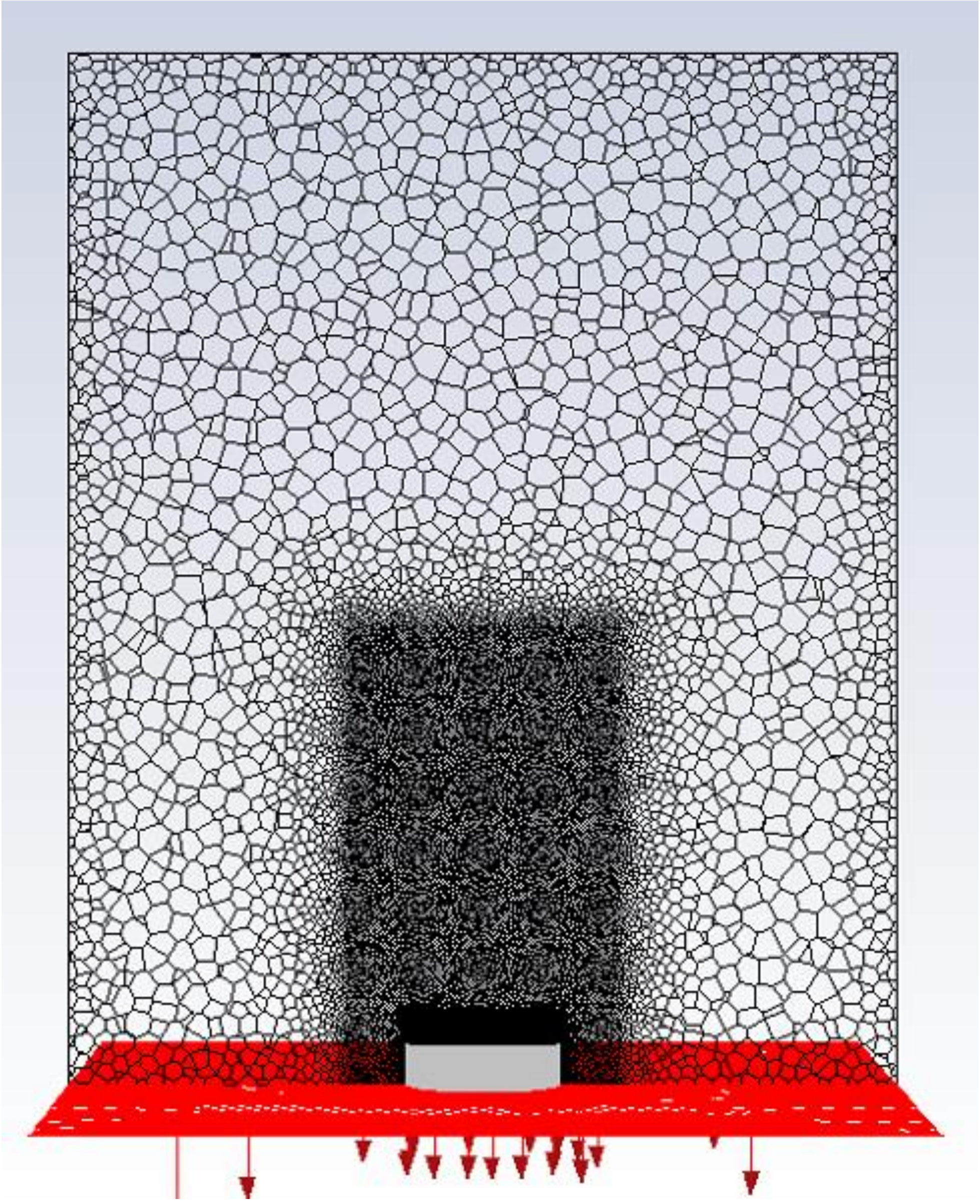
- Smaller domain (1 x 1 x 1.5 m)
- Basic EDM
- Not converged for steady and transient
- Issues at the pressure outlets
- Need of a larger domain



Results –Bigger domain

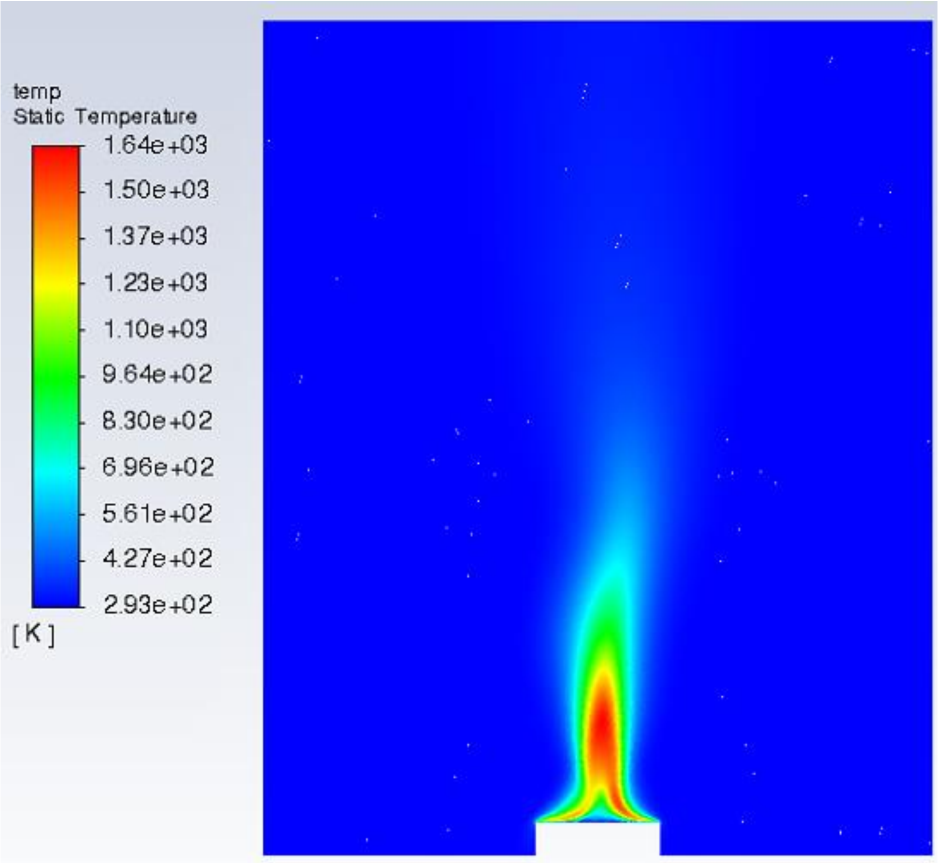
- Domian (2 x 2 x 2.5 m)
- Refined mesh around the flame area and inlet – 2 mil. cells



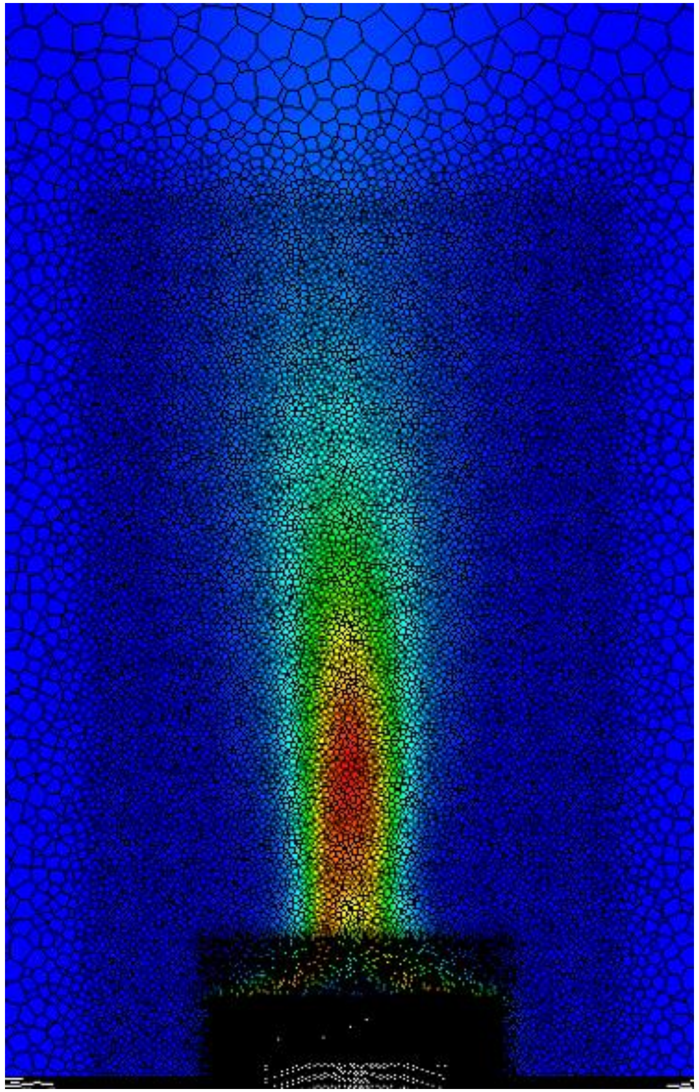


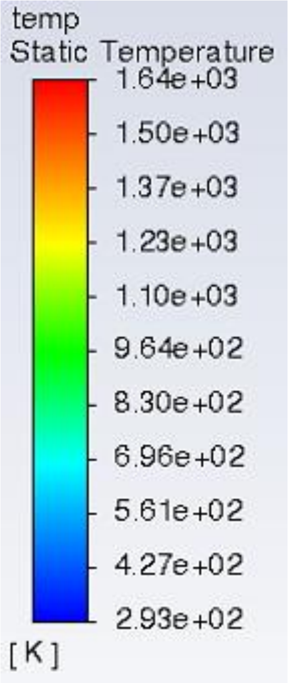
Results –Bigger domain

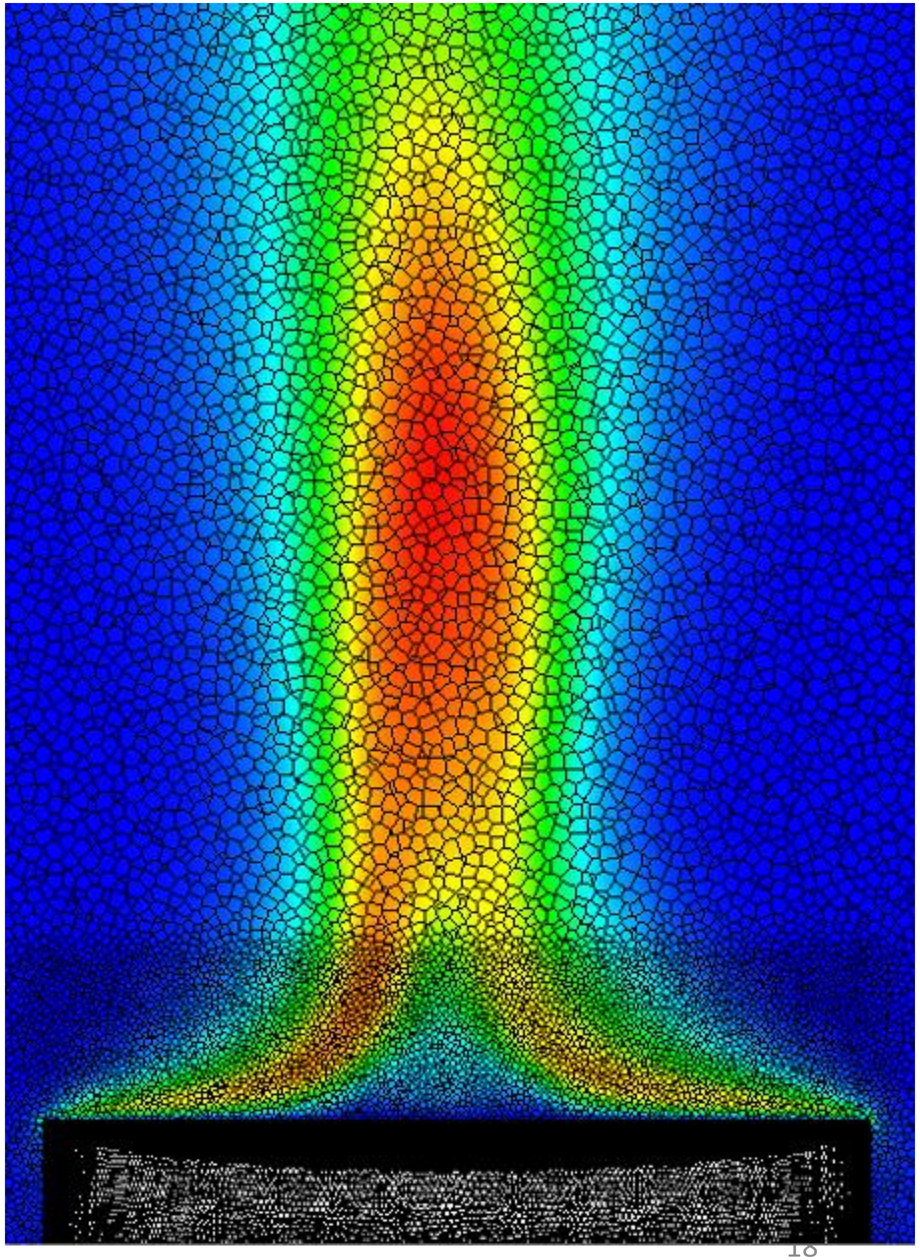
- EDM for combustion, steady state, RANS
- Not completely steady
- Boundary conditions are correct
- Stepping stone to complicated simulations



Results –Bigger domain

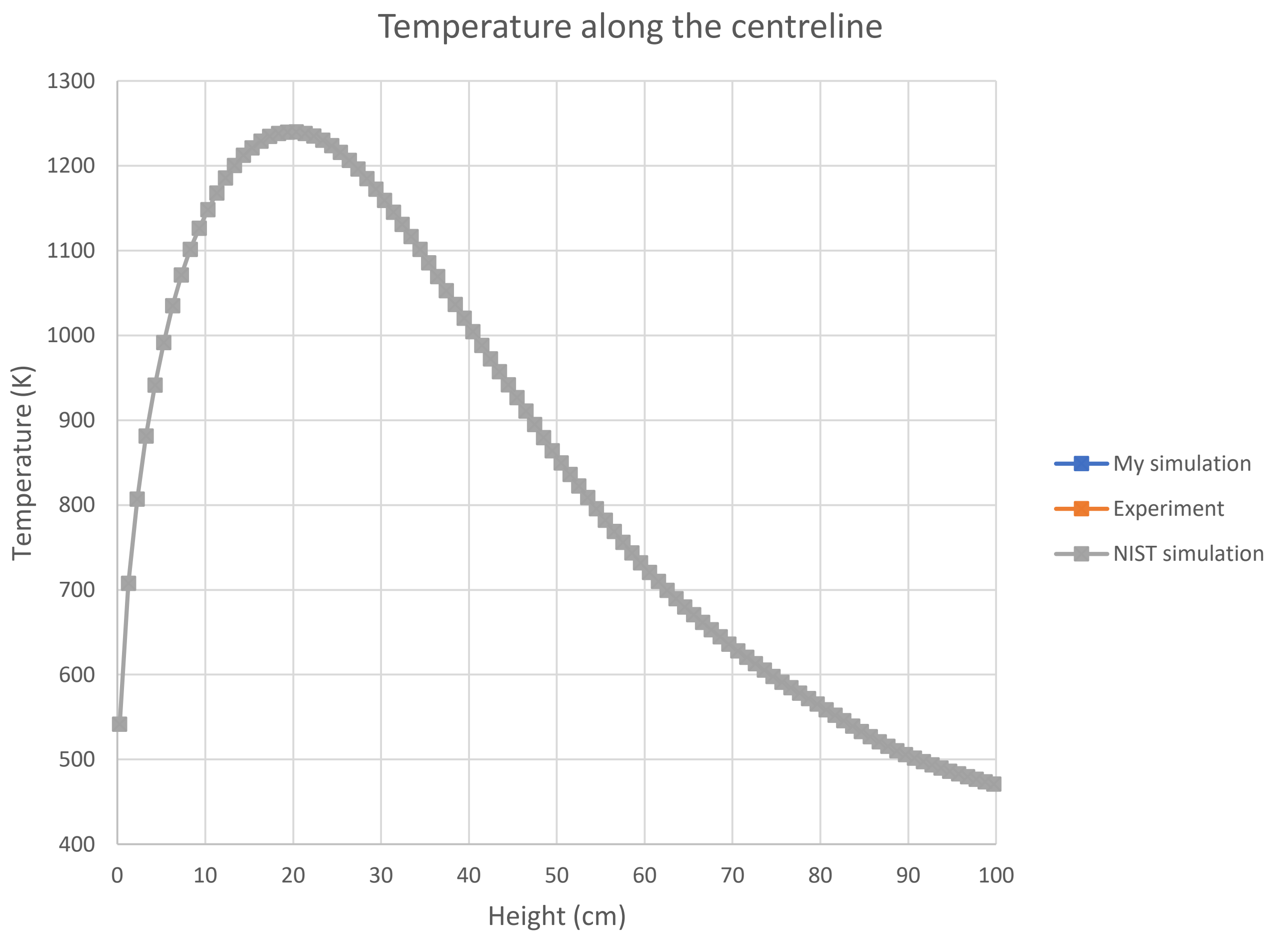






Comparison with experimental data

- Simple simulation – good result
- Overpredicted temperature
  - Complete combustion
- Developing this simulation can lead to good results



What to do next?

- Transient simulations with RANS and LES
- Adding creation of CO
- Use of Conserved scalar combustion model
- Creating a mesh size independence study

- Guidelines for using Fluent in fire engineering problems
- Comparing available models

Bibliography

[1]: Hurley, Morgan J., et al., eds. *SFPE handbook of fire protection engineering*. Springer, 2015.

[2]: Hamins, A. , Maranghides, A. , McGrattan, K. , Ohlemiller, T. and Anleitner, R. (2005), Experiments and Modeling of Multiple Workstations Burning in a Compartment. Federal Building and Fire Safety Investigation of the World Trade Center Disaster (NIST NCSTAR 1-5E), National Construction Safety Team Act Reports (NIST NCSTAR), National Institute of Standards and Technology, Gaithersburg, MD, [online], https://tsapps.nist.gov/publication/get_pdf.cfm?pub_id=101033 (Accessed November 21, 2021)

[3]: Yeoh, Guan Heng, and Kwok Kit Yuen. *Computational fluid dynamics in fire engineering: theory, modelling and practice*. Butterworth-Heinemann, 2009.

[4]: Som, S., P. Senecal and E. Pomraning. "Comparison of RANS and LES Turbulence Models against Constant Volume Diesel Experiments." (2012).

Thank you!

Any comments, questions, advices, tips?

This work was supported by the grant of Specific university research - grant No. A1-FCHI_2021_004