4. Computational Fluid Dynamics

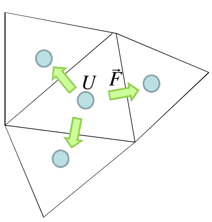
Dr. Gergely Kristóf
Dept. of Fluid Mechanics, BME
February, 2009.

Principles of CFD

- Our aim is the approximate solution of the governing equations via numerical methods.
- Leading methods:
  - **Finite volume method**; - prevails in CFD
  - Finite difference method;
  - Finite element method.
- Some less widely spread methods:
  - Spectral methods;
  - Mesh-less methods;
  - Latice-Boltzmann.
- The domain is subdivided into smaller volumes (cells) in which the solution is approximated by simple functions (e.g. by linear functions). The process of subdivision is called: grid generation or meshing.
- The approximate solution is based on discrete values of the field variables stored in specific points of the numerical grid.
- The interaction between the meshed domain and the outer world is specified in the form of boundary conditions over the contour surface of the domain.

Finite volume method

The generic transport equation



U: volume intensity of a conserved quantity

$$\frac{\partial}{\partial t}\int_V U\,dV + \oint_A \vec{F}\cdot d\vec{A} = \int_V S_V\;dV + \oint_A \vec{S}_A\cdot d\vec{A}$$

Conserved quantity per unit mass of fluid:

$$\Phi = U / \rho$$

Convective and conductive (diffusive) fluxes:

$$\vec{F}_C = \rho\,\vec{v}\,\Phi \qquad \vec{F}_D = -\Gamma\nabla\Phi$$

$$\frac{\partial}{\partial t}\int_V \rho\,\Phi\,dV + \oint_A \rho\,\Phi\,\vec{v}\cdot d\vec{A} = \oint_A (\Gamma\nabla\Phi + \vec{S}_A)\cdot d\vec{A} + \int_V S_V\;dV$$

The generic transport equation in differential form:

conductive flux

$$\frac{\partial \rho\phi}{\partial t} + \nabla\cdot(\rho\phi\vec{v}) = \nabla\cdot\vec{S}_A + \nabla\cdot(\Gamma\nabla\phi) + S_v$$

convective flux

Transport equ.	ϕ
Continuity	1
x-momentum	u
y-momentum	v
z-momentum	w
Stagnation energy (internal+chinetic)	e

Conservative form of the governing equations for single phase laminar flow:

$$\frac{\partial \rho}{\partial t} + \nabla\cdot(\rho\vec{v}) = S_m$$

$$\frac{\partial \rho u}{\partial t} + \nabla\cdot(\rho u\vec{v}) = -\frac{\partial p}{\partial x} + \nabla\cdot(\mu\nabla u) + \rho g_x + S_u$$

$$\nabla\cdot(-p\underline{E} + \underline{\tau})$$

$$\frac{\partial \rho v}{\partial t} + \nabla\cdot(\rho v\vec{v}) = -\frac{\partial p}{\partial y} + \nabla\cdot(\mu\nabla v) + \rho g_y + S_v$$

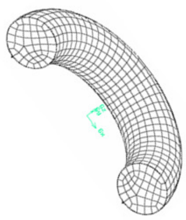
$$\frac{\partial \rho w}{\partial t} + \nabla\cdot(\rho w\vec{v}) = -\frac{\partial p}{\partial z} + \nabla\cdot(\mu\nabla w) + \rho g_z + S_w$$

(Most turbulence models change only the transport coefficients λ and μ.)

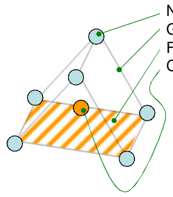
$$\frac{\partial \rho e}{\partial t} + \nabla\cdot(\rho e\vec{v}) = \nabla\cdot(-p\vec{v} + \underline{\tau}\cdot\vec{v}) + \nabla\cdot(\lambda\nabla T) + \rho\vec{g}\cdot\vec{v} + S_e$$

Numerical grid

Numerical grid



Elements of the numerical grid



Cell: Control volume. Field variables are (typically) defined in cell centroids.
Node: Grid geometry is defined by node positions.
Grid lines: Straight sections between neighboring nodes.
Facets: Cell sides defined by 3 or 4 nodes. A cell can have arbitrary number of sides.

Characteristics of the Finite Volume Method (FVM)

- The governing equations are used in **integral form**. (Integrated over cell volumes.)
- Divergence terms are converted into **surface integrals** over the facets enclosing the cells. The numerical approximation of the flux integral for one facet depends only on two unknown ϕ values stored in the centers of the two neighboring cells adjacent to the facet.
- As a result of this - so called discretization - process, every transport equation provides one (non-linear) **algebraic equation per cell**, e.g. if we have 5 transport equations and 1 000 000 cells, then we obtain a system of 5 000 000 non-linear algebraic equations. In the case of time dependent problems, we have to solve this system of equations in every time step.
- Each algebraic equation contains unknown ϕ values for **one** particular **cell and for all of its neighboring cells**. This is e.g. 5 unknowns per equations for tetrahedral grids.
- Due to the large number of unknowns and the non-linearity of the system of equations, **iterative** methods have to be used. The solution is first **initialized**, and then iteratively refined, thus **converging** towards the final solution.
- Integrals of fluxes over the boundary facets need to be defined in consistence with the physical characteristics of the region outside of the boundary, done by imposing additional mathematical conditions: **boundary conditions**.
- Surface integrals are numerically evaluated for every small facet, such as for that connecting two neighboring cells. These integrals express the flow rates of conserved quantities (mass, momentum, energy). When we calculate the integrals for such conserved quantities of the whole domain, the surface integrals for the internal facets are canceled, therefore the conservation equations for the whole domain are exactly fulfilled. This is called the **conservative behavior** of the finite volume method.

Overview of the process

1. Creation of model geometry,
2. Meshing,
3. Marking the boundary zones,
4. Selection of physical model, specification of material properties.
5. Parameterization of BC-s,
6. Adjustment of numerical controls
7. Initialization
8. Iteration
9. Visualization of the results

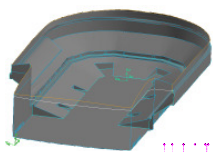
Development of User Defined Functions, if necessary.

Optimization loop

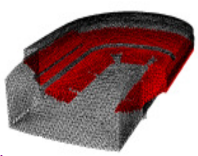
Application examples

1. HV&AC analyses of a sport hall

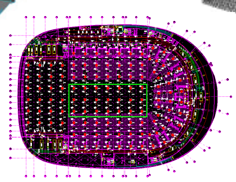
Papp László Sportaréna, Budapest, 2001.



Geometrical model



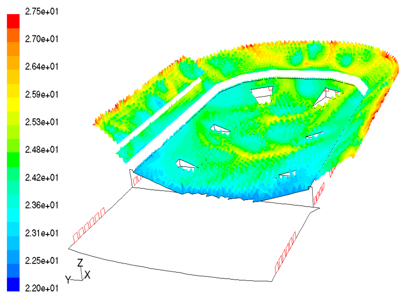
Mesh 40 k cells, 2m resolution



Air inlets (red dots)

Department of Fluid Mechanics, BME

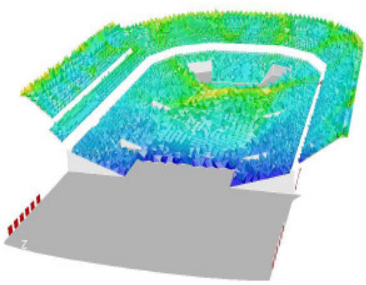
Air temperature



Department of Fluid Mechanics, BME

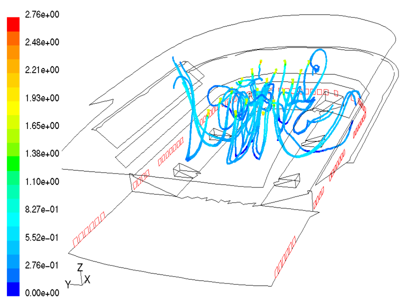
Humidity

Moisture load: 0.0187 g/s/person



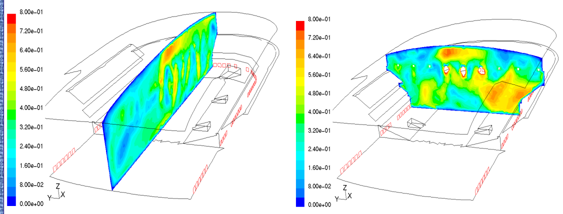
Department of Fluid Mechanics, BME

Streamlines in concert operation mode



Department of Fluid Mechanics, BME

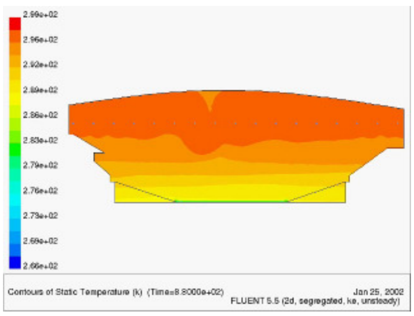
Contour plots of velocity magnitude



Department of Fluid Mechanics, BME

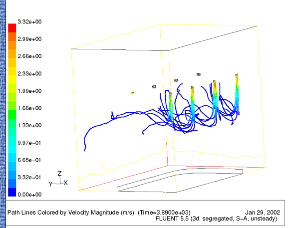
Temperature transient during start-up

Total time frame: 25 minutes

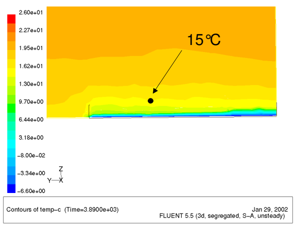


Department of Fluid Mechanics, BME

Fulfillment of a sports related technical requirement: temperature above the ice



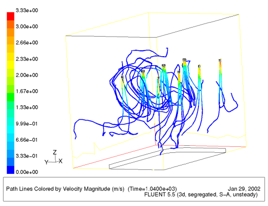
1 row of air inlets operating in jet mode



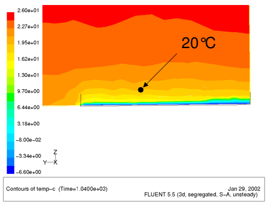
Temperature distribution

Department of Fluid Mechanics, BME

Fulfillment of a sports related technical requirement: temperature above the ice



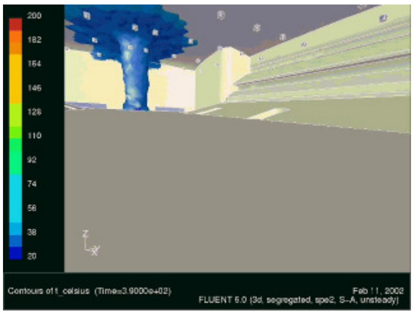
3 rows of air inlets operating in jet mode



Temperature distribution

Smoke analyses

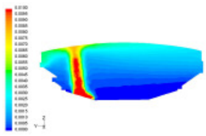
at 10 minutes

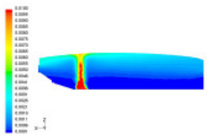


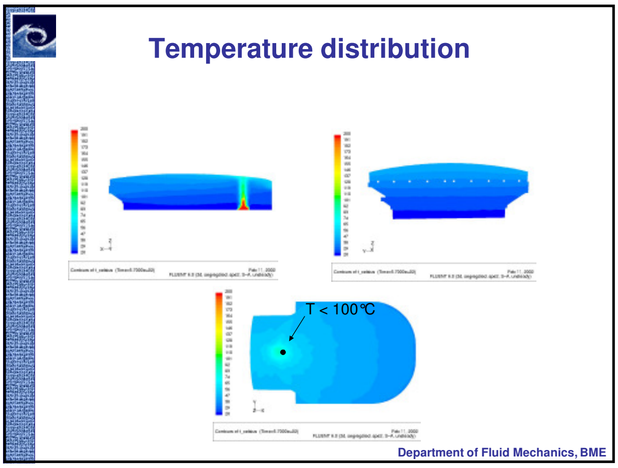
The fire spreads with time

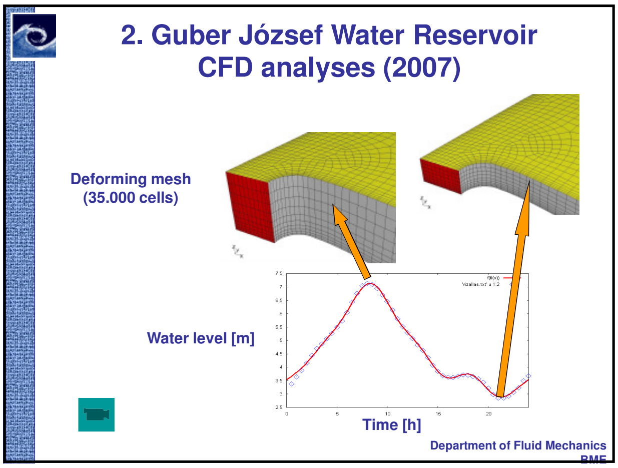
Different fire locations

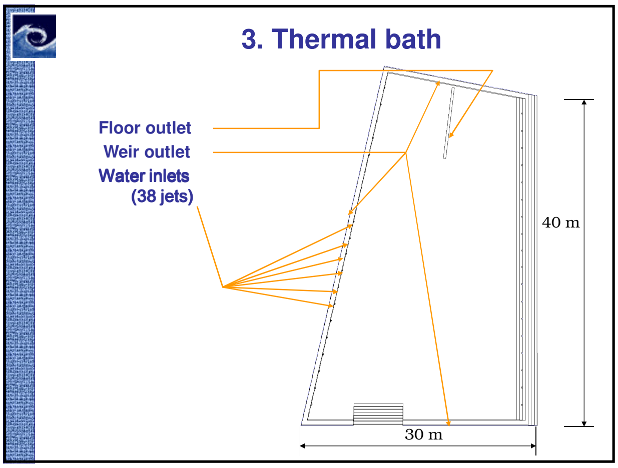
CO_2 concentration



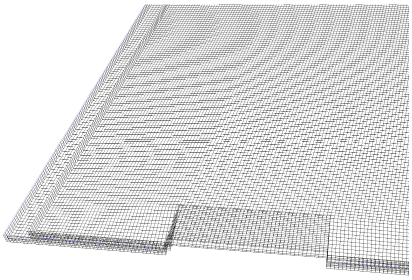



Temperature distribution
T < 100°C
Department of Fluid Mechanics, BME


2. Guber József Water Reservoir
CFD analyses (2007)
Deforming mesh
(35.000 cells)
Water level [m]
Time [h]
Department of Fluid Mechanics
BME


3. Thermal bath
Floor outlet
Weir outlet
Water inlets
(38 jets)
40 m
30 m

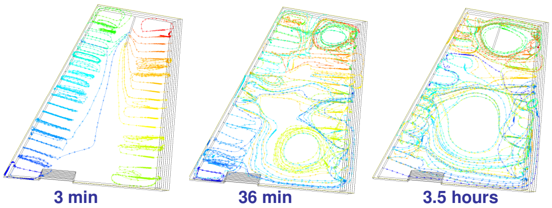
Numerical mesh



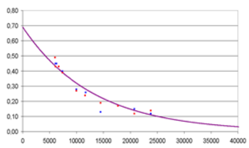
Spatial resolution: 0,14 x 0,14 x 0,14 m^3

Number of cells: 470 000

Flow structure



3 min **36 min** **3.5 hours**

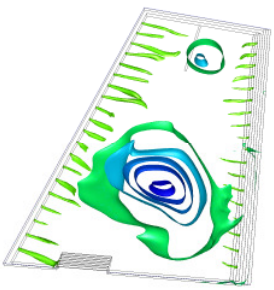


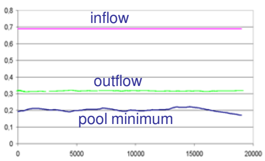
Modeled as
a volume source:

$$S_{Cl} = \frac{c_{Cl}}{\tau}$$

Department of Fluid Mechanics BME

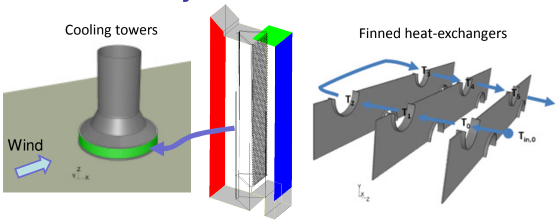
Evolution of the active chlorine concentration





Department of Fluid Mechanics BME

Multiscale modeling of flows in finly structured domains



Sometimes the microscale geometry is periodic, but the macroscopic flow is aperiodic. **You can create a microscale model with periodic BCs, and run it with different boundary conditions taken from the macroscale model.**

In other cases you can assume geometrical periodicity…