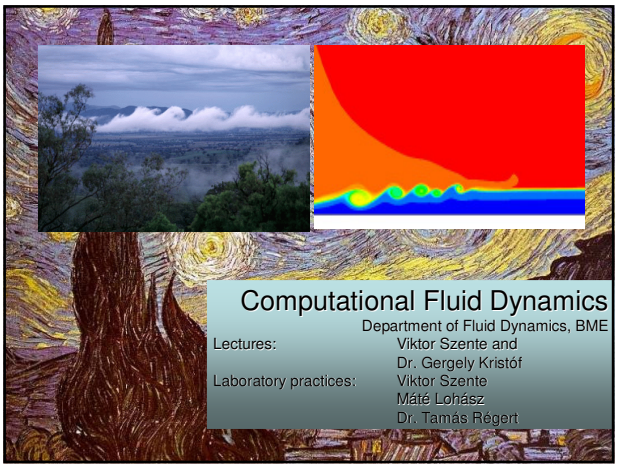
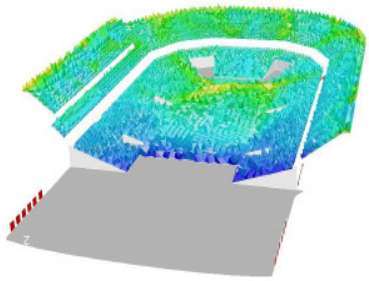


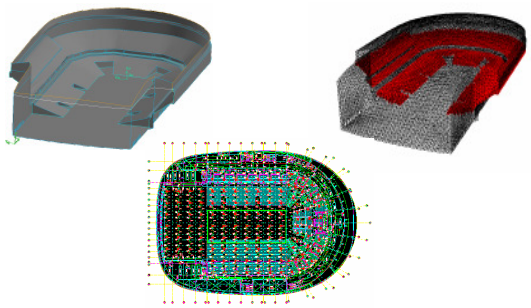
Humidity

Moisture load: 0.0187 g/s/person



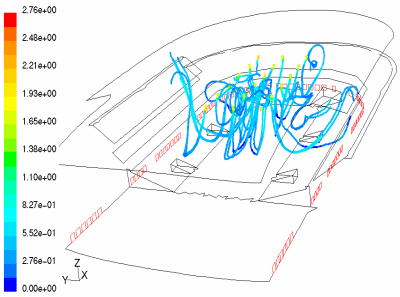
HV&AC analyses of a sport hall

3D geometrical model of the internal space and the stands
Number of cells 339000, spatial resolution 2m

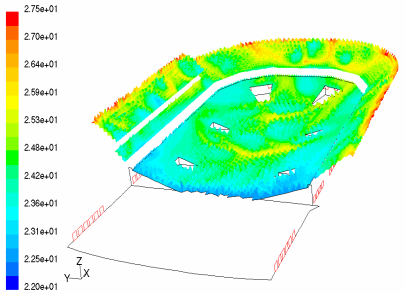


Air inlets (red dots)

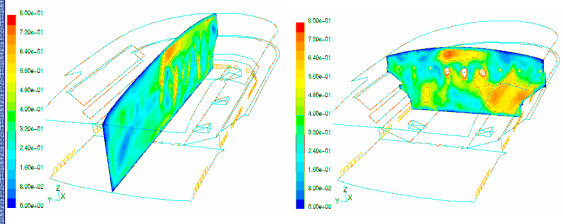
Streamlines in concert operation mode



Air temperature

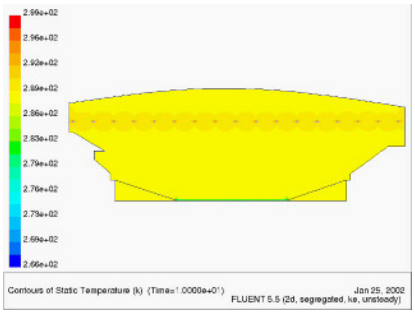


Contour plots of velocity magnitude



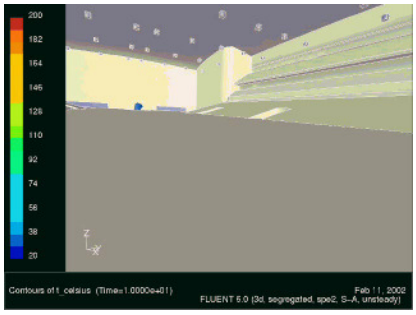
Temperature transient during start-up

Total time frame: 25 minutes



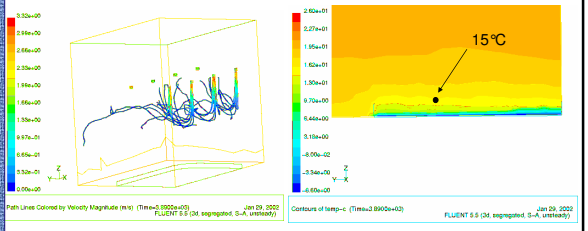
Smoke analyses

a 10 minutes scenario



Spatial extension of the fire is increased by time

Fulfillment of a sport-technical requirement: temperature above the ice

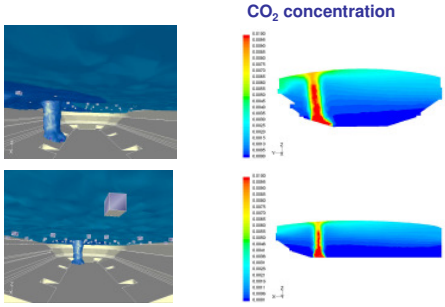


1 row of air inlets operate in jet mode

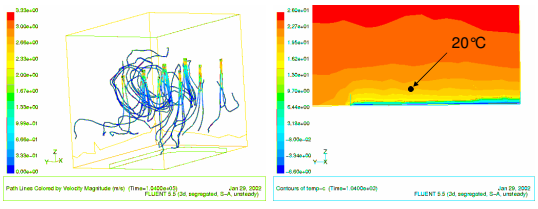
Temperature distribution

Different fire locations

CO_2 concentration



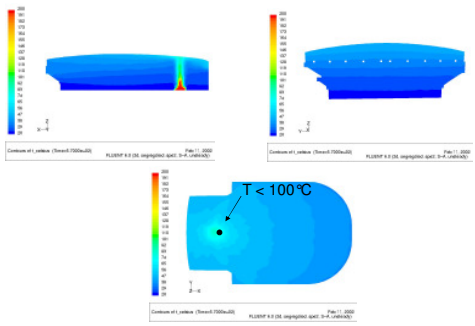
Fulfillment of a sport-technical requirement: temperature above the ice



3 rows of air inlets operate in jet mode

Temperature distribution

Temperature distribution



About the course

- Teaching targets:
  - To provide practical knowledge in CFD using modern CFD tools:
    for 0D and 1D problems: AmeSim;
    for 2D and 3D problems: ANSYS-FLUENT.
  - To deliver essential theoretical knowledge for trustable CFD modeling.
- Practices are in HSzK laboratories. Course material (presently only in Hungarian):
  http://www.ara.bme.hu/~cfd/FLUENTkurzus/Index.htm
  Please note the URL! (Case sensitive.)
- The first 6 laboratory practices are **guided**.
- The **tests** will be on 14. week (last week) during the lecture and the practices. 35 points are given for successful practical tests and 15 points for theoretical a test. Maximum 50 point can be collected from this half of the subject, the final mark is calculated from the summed result of the two half semesters.

Finite volume method

- 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 numeric approximation of the flux integral for one facet depend 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 equation in every time step.
- Each algebraic equations contain unknown ϕ values from one particular cell and from all neighboring cells. This is e.g. 5 unknowns per equations for tetrahedral grids.
- Due to the high number of unknowns and the non-linearity of the system of equations **iterative** solution methods have to be used. The solution is first **initialized**, and than, iteratively refined thus **converging** towards the final solution.
- Integral of fluxes over the boundary facets need to be defined in consistence with the physical characteristics of the region outside of the boundary by imposing additional mathematical conditions: **boundary conditions**.
- Surface integrals are numerically evaluated for every small facets, such as for those connecting two neighboring cells. These values express the flow rates of conserved quantities (mass, momentum, energy). When we calculate the integral of such conserved quantities for the whole domain, the surface integrals for internal facets are canceled, therefore the conservation equations for the whole domain are exactly fulfilled. This is called as the **conservative behavior** of the finite volume method.

Principles of CFD

- Our aim is the approximate solution of the governing PDE-s via numerical methods.
- Four major methods are in use:
  - finite difference method,
  - finite element method,
  - spectral methods,
  - finite volume method.
- In the field of CFD finite volume methods prevail.
- The domain is subdivided into smaller volumes (cells) in which the solution is approximated by more simple functions (e.g. by linear functions).
- The process of subdivision is called: grid generation or meshing.
- Interaction between the meshed domain and the outer world is specified in the form of boundary conditions at the contour surface of the domain.

Overview of the process

1. Creation of model geometry,
2. Meshing,
3. Marking the boundary zones,

pre-processing
in GAMBIT
.dbs, →.msh

4. Selection of physical model, specification of material properties.
5. Parameterization of BC-s,
6. Adjustment of numerical controls
7. Initialization
8. Iteration

solution
in FLUENT
*.cas, *.dat

9. Visualization of the results

post-processing
in FLUENT

This workflow will be demonstrated by the rounded orifice example...

The generic transport equation in differential form:

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

conductive flux

convective flux

Transport equ.	ϕ
Continuity	1
x-momentum	u
y-momentum	v
z-momentum	w
Energy	e

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

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

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

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

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

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

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