Reducing Air Toxic Impact from Power Plants Startups through CFD-Assisted Design of Chimneys

Paolo Zannetti, The EnviroComp Institute
zannetti@envirocomp.org
Giuseppe Bucci, SINAPSI Innotec Srl
pino.bucci@sinapsi.net

Topics

1. The problem of frequent startups in power plants
   ➔ potential adverse effects on ground concentrations: lower plume temperature ➔ lower plume rise ➔ higher concentration
2. Computational Fluid Dynamics (CFD) methods
   ➔ simulation of gasses inside the chimney
3. Future use of CFD to estimate expected improvements e.g., from lining systems inside the chimney ➔ faster reach of steady state plume

1. Frequent Startups in Power Plants

- Expansion of intermittent wind and solar power needs backup from power plants
- A combination of renewable and fossil technologies seems necessary
  - Allow a precise overlap between the power production curve and the demand peak
- ➔ Frequent startups of power plants
- Startup conditions ➔ more ground-level pollution
  - Because of lower plume temperature ➔ lower plume elevation

Power Plant Plumes

- If a Power Plant operates in steady state, stack emissions are relatively constant
- But during startups it may take a few hours (ΔH) to reach steady state ➔ maximum exit temperature
- During ΔH stack exit temperatures will be lower ➔ higher ground-level concentrations
- Plume dynamics inside a stack can be simulated using CFD methods

denity

2. Computational Fluid Dynamics (CFD)

CFD analysis is an effective tool for design support in many sectors. It is used for product optimization or to design plant components where fluid flow and heat transfer are key phenomena in:

Aerospace industry

Aeronautics

Automotive

Energy industry

Chemical plants

Electronics

Pharmaceutics, etc.

denity

CFD

- Computational Fluid Dynamics, known as CFD, is based on numerical techniques for solving the well-known Navier Stokes equations,

plus additional equations for energy conservation and turbulence

Continuity Equation

$$\nabla \cdot \vec{V} = 0$$

Momentum Equations

$$\rho \frac{D\vec{V}}{Dt} = -\nabla p + \rho \vec{g} + \mu \nabla^2 \vec{V}$$

Total derivative

=

$$\rho[\frac{\partial V}{\partial t} + (V.\nabla)V]$$

Change of velocity with time

Convective term

Pressure gradient

Fluid flows in the direction of largest change in pressure.

Body force term

External forces, that act on the fluid (gravitational force or electromagnetic).

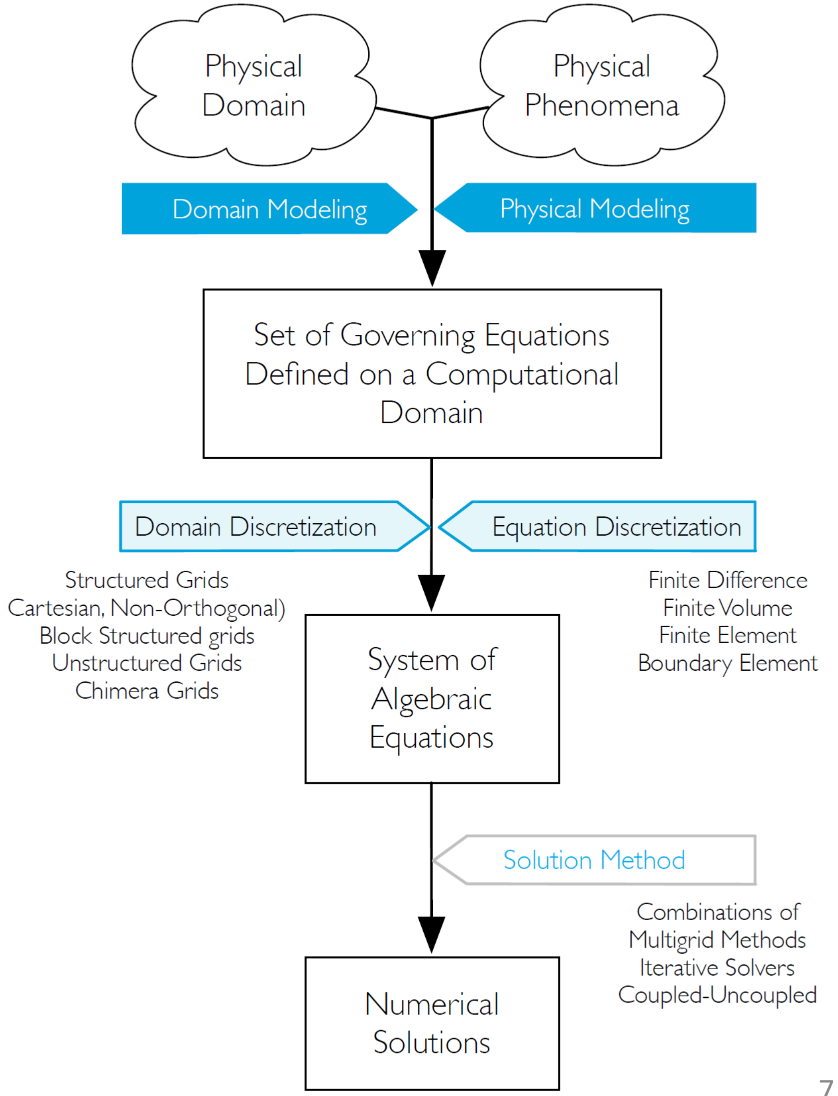
Diffusion term

For a Newtonian fluid, viscosity operates as a diffusion of momentum.

CFD

The PDE of fluid dynamics require to be solved by numerical methods based on discretisation of domain and equations.

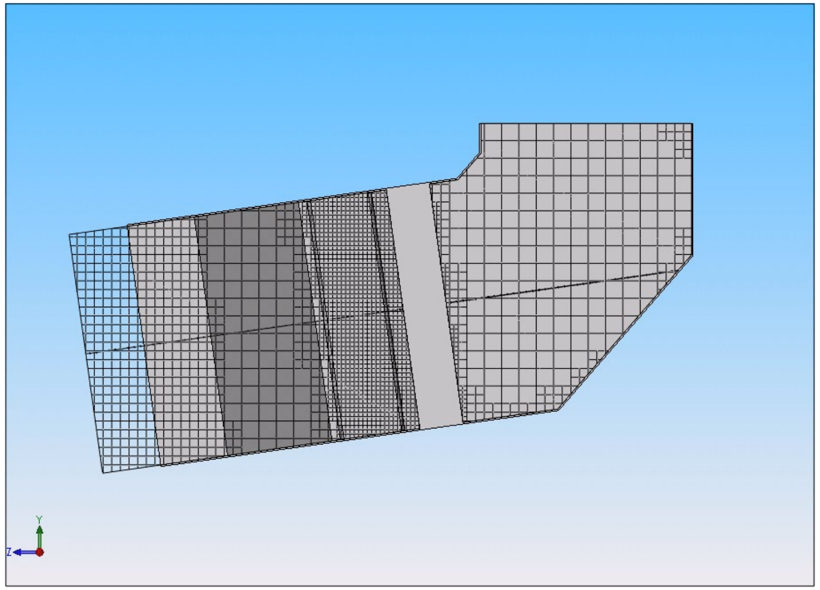
CFD works by the following typical scheme



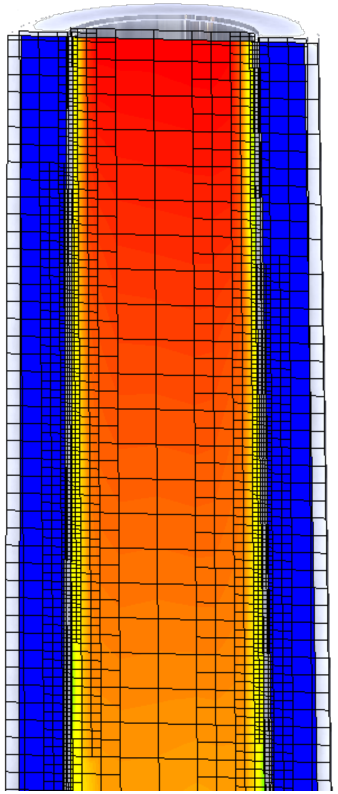
denity

CFD

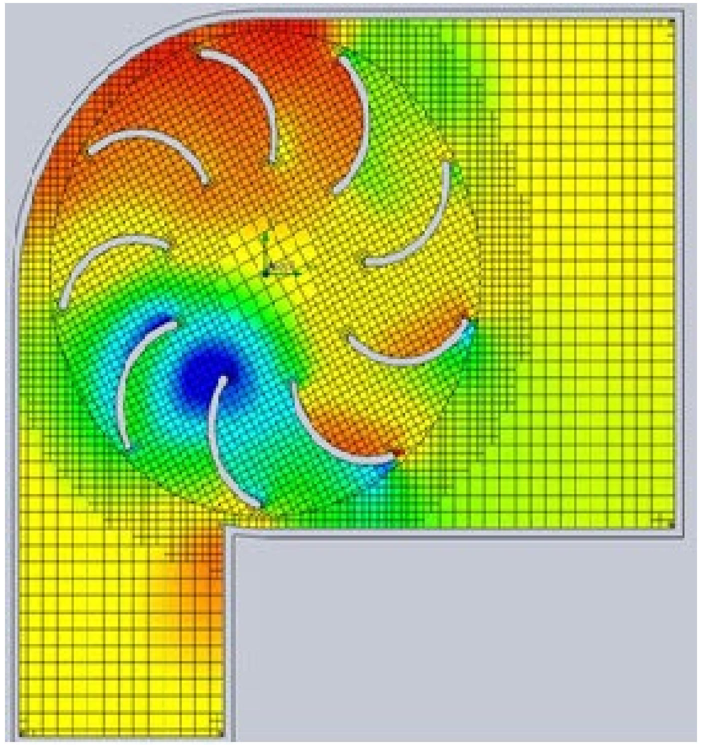
Examples of «Discretization»



Silencer – Section view



Chimney stack



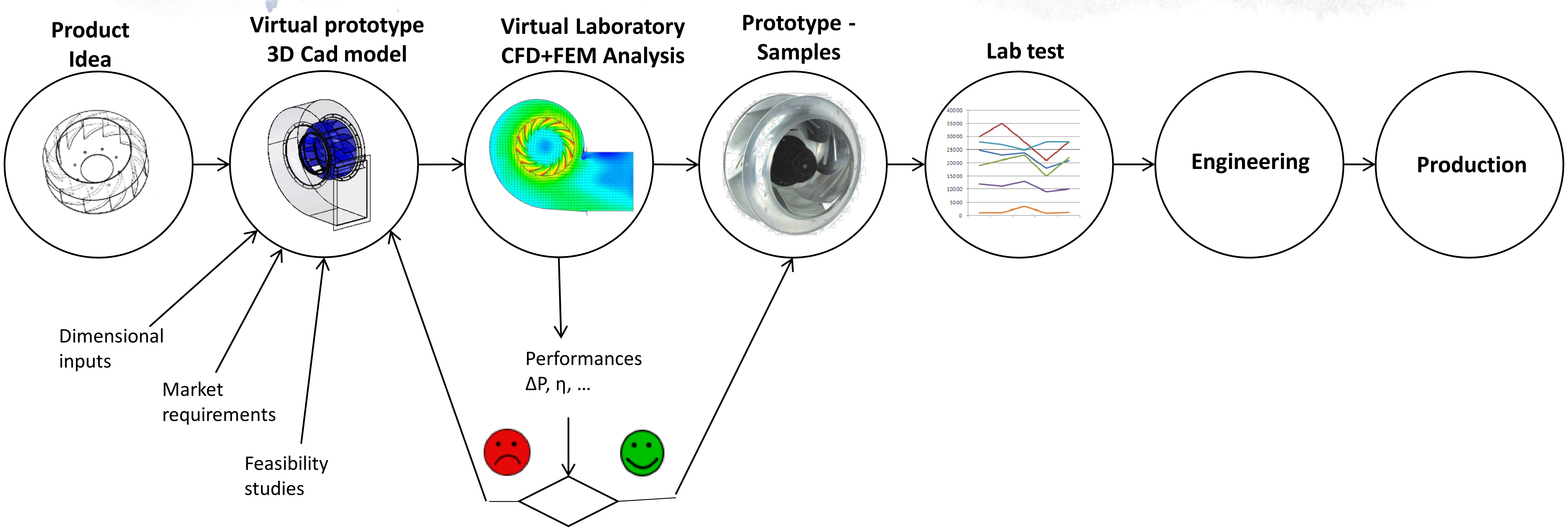
Impeller

CFD advantages

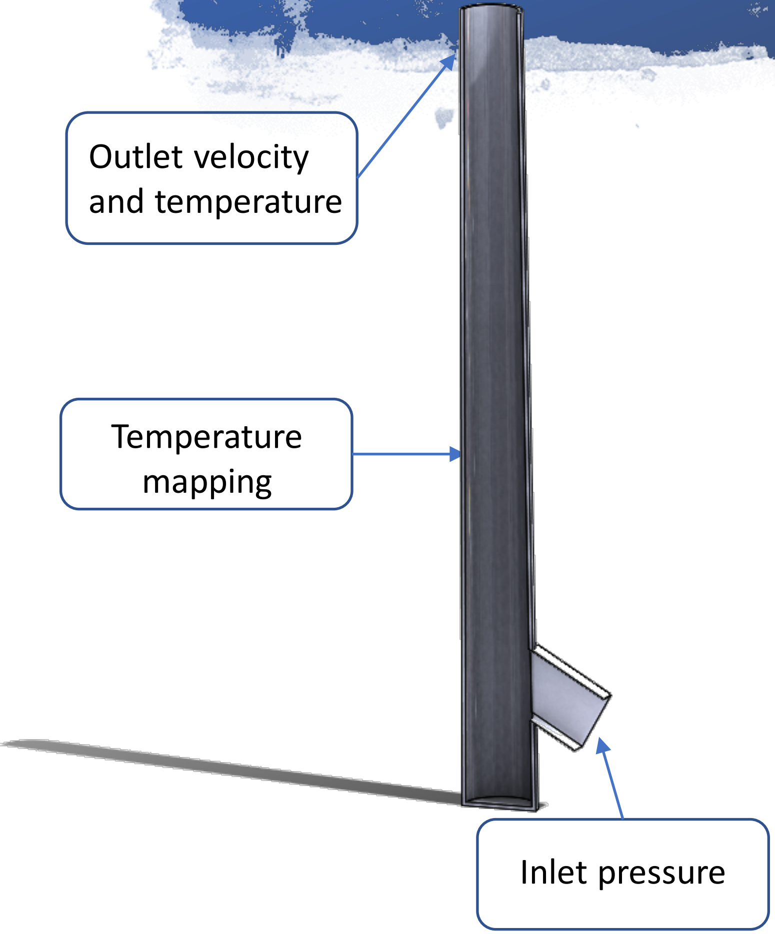
CFD is now a fundamental tool in the R&D and Engineering departments for many reasons that can be summarized as follows:

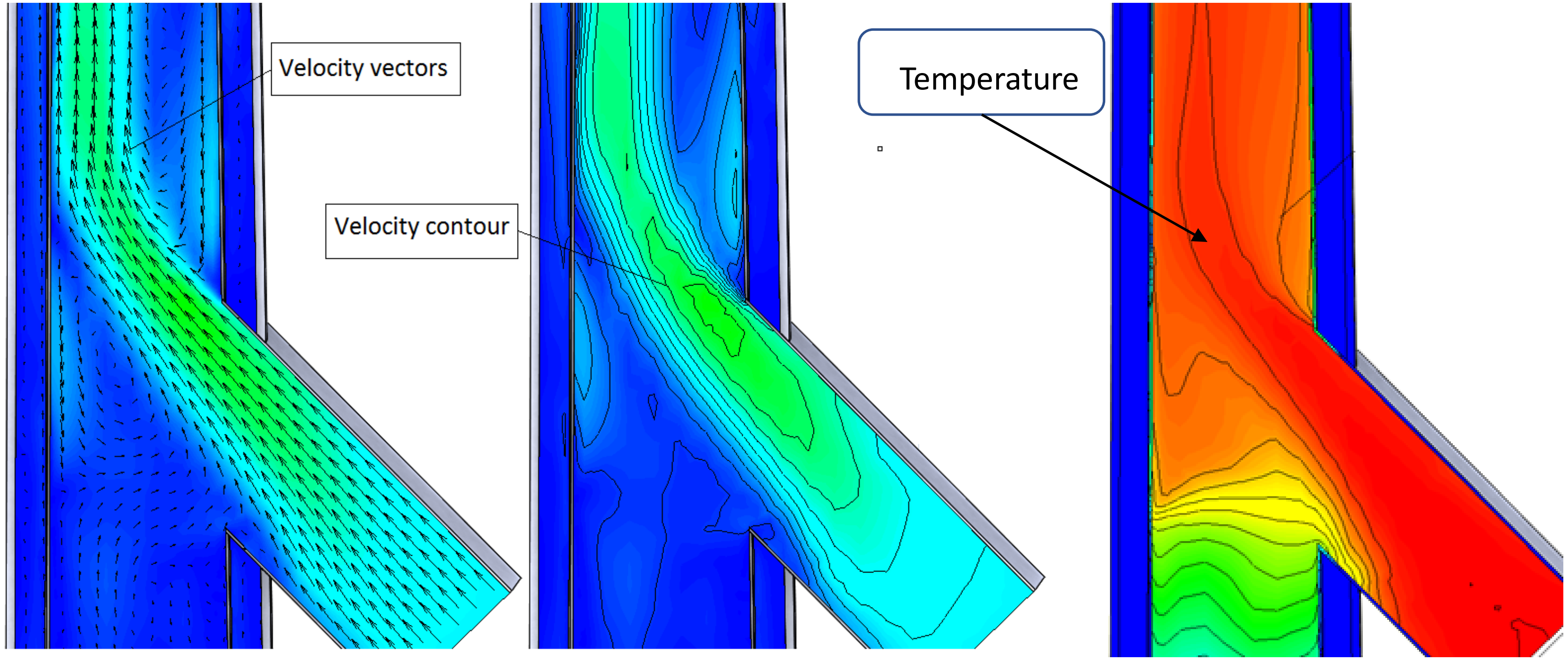
- Allows to check the ideas to be developed still in the initial stage of design
- Reduces the costs of prototyping and laboratory tests,
- Provides a clear picture of the phenomena developing inside components, allowing to predict and to better understand the results of a project configuration and finally improve its development

CFD SIMULATION TOOLS
Typical Design flow process in R&D department

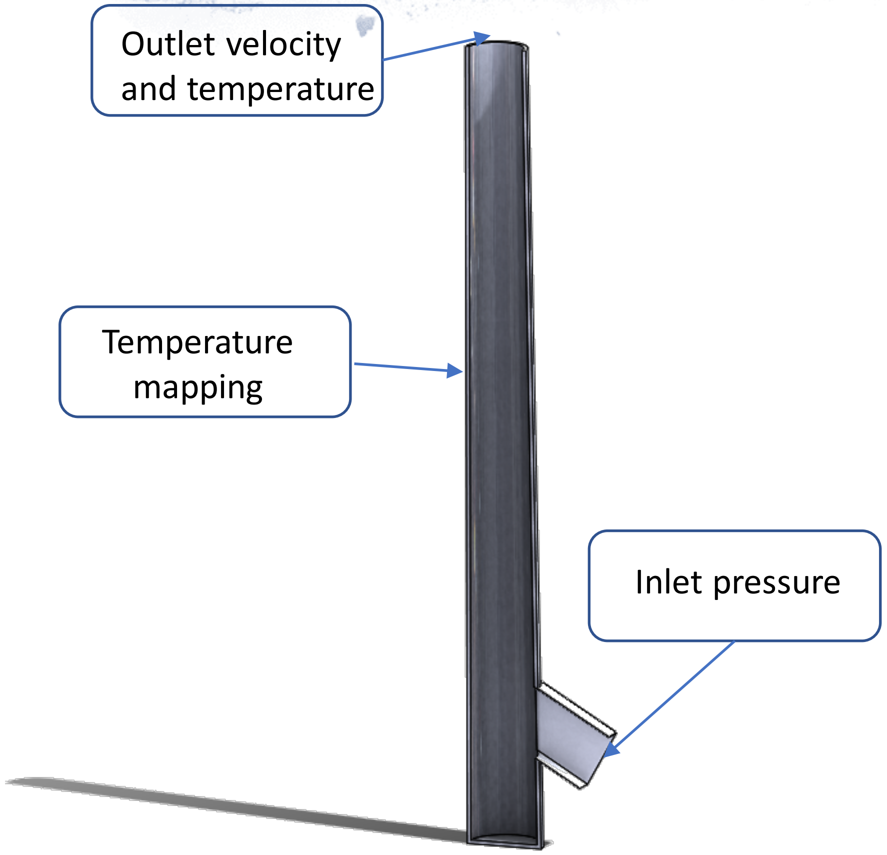


CFD Analysis in chimney stack design
The output results



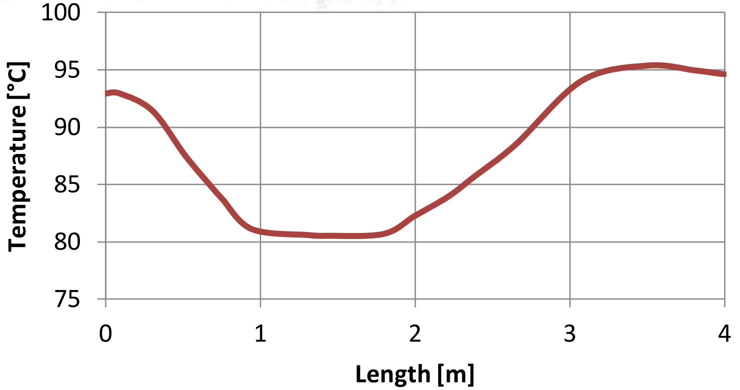


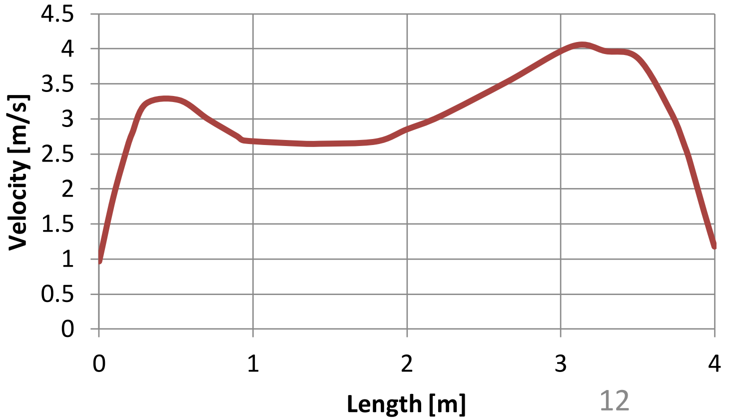
CFD Analysis in chimney stack design
The output results



Streamlines

Temperature and velocity profiles





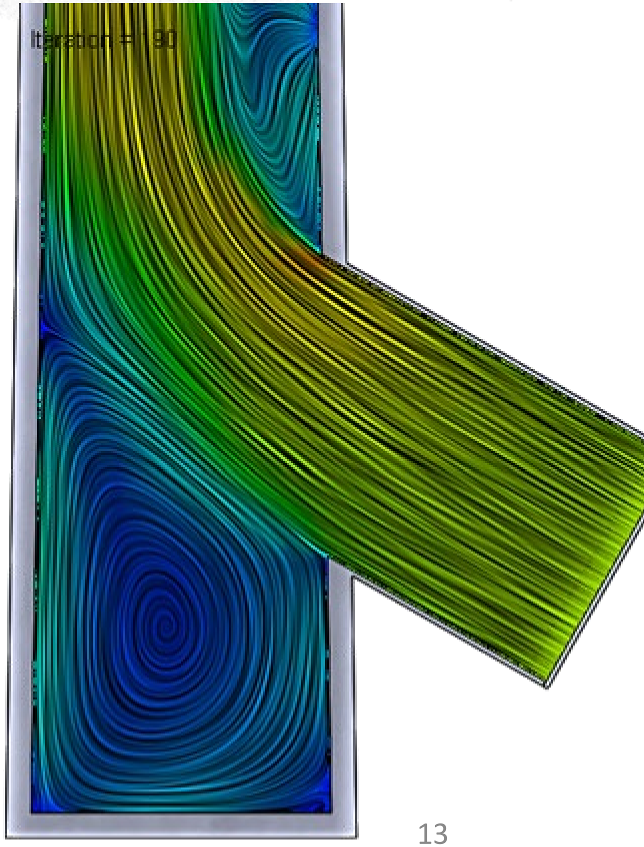
CFD in chimney stack design

Flow analysis in the inlet area to study:

- inlet angle, cross area shape and dimensions

to minimize pressure drops

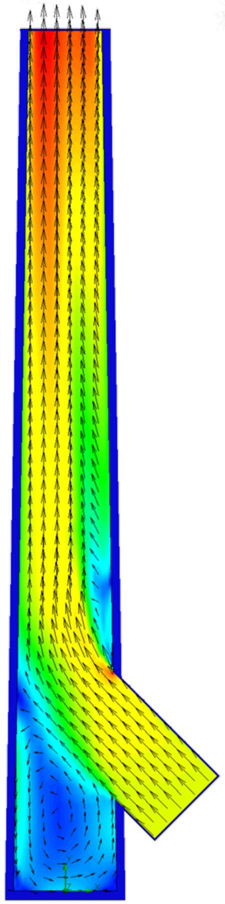
- Stagnation area under the inlet duct
- Impact and erosion problems
- Non uniform temperature distribution and related stress

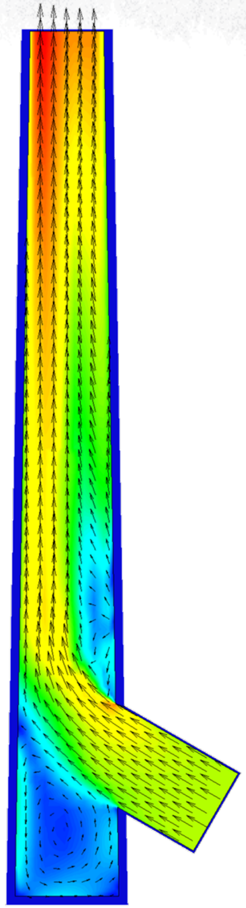


CFD in chimney stack design

Flow analysis in the inlet area

Example of two different inlet angle





CFD in chimney stack design

Flow analysis in the chimney:

- Accurate estimate of pressure drops
- Calculation of the temperature distribution in the wall
- Correct estimate of the thermal resistance of the wall and heat fluxes also when there are air gaps

CFD in chimney stack design

Flow analysis at the outlet

- Calculation of velocity and temperature of gases as an input for Air Pollution Models
- Study of flow outside the chimney. In this case it is possible to extend the domain around and down wind the chimney, but the size of the numerical problem and time calculation grow up

CFD in chimney stack design

Transient analysis gives information about:

- Temperature maps for stress analysis
- Pressure variations and waves during discharge events
- Velocity and Temperature at the outlet for Air Pollution Modeling

CFD in chimney stack design

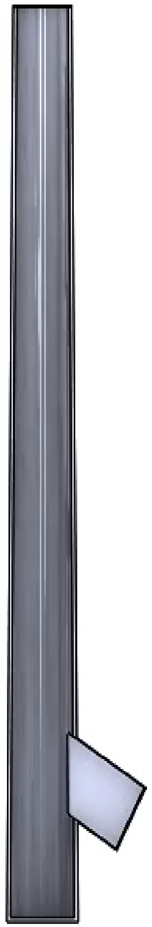
Example of Transient analysis:

Chimney Height 100 m

Inner diameter at top 7 m

Inlet velocity, ramp up 1 m/s to 10 m/s

Inlet temperature, ramp up 50 to 100°C



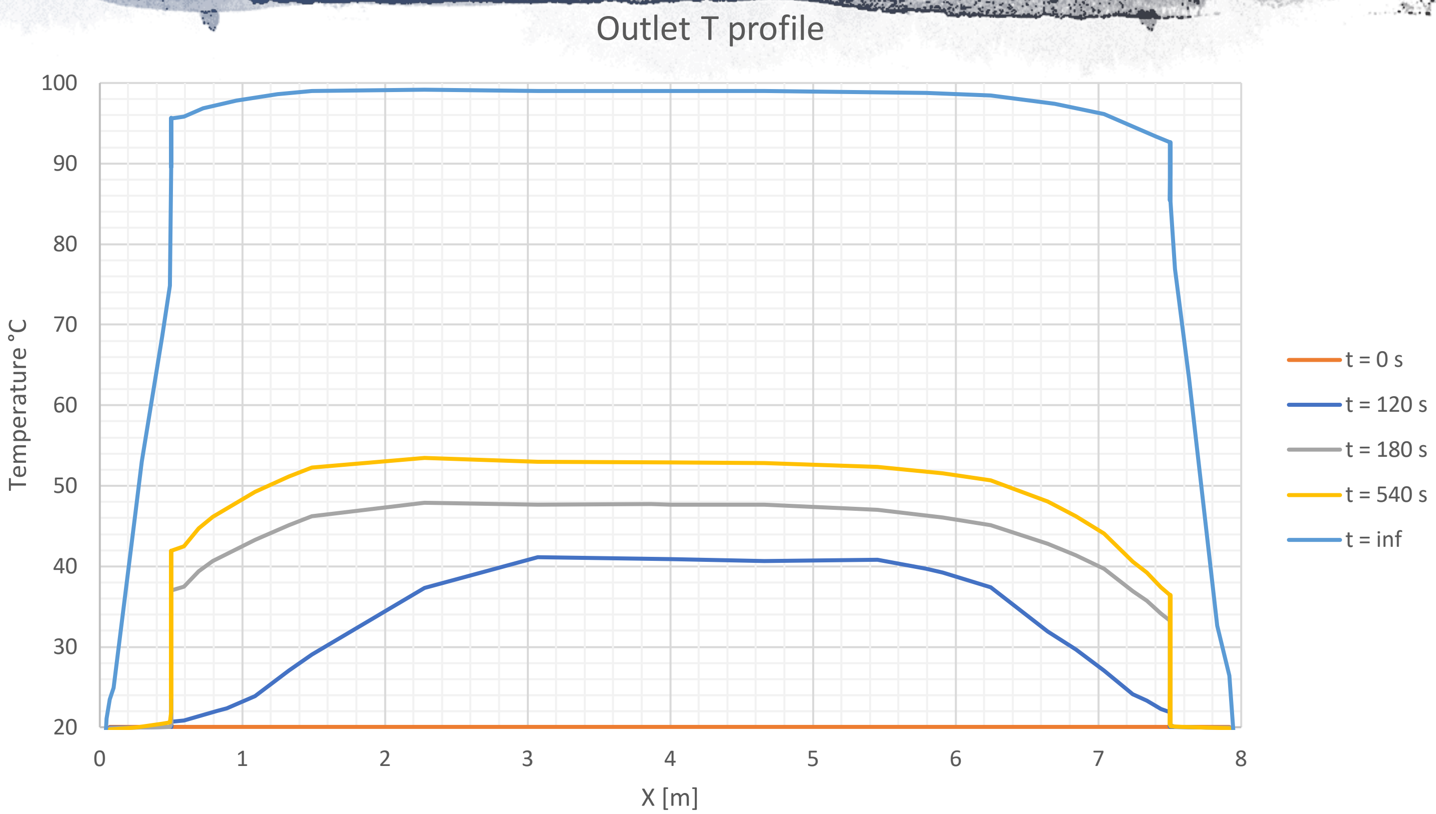
Temperature of flue gases variation during time

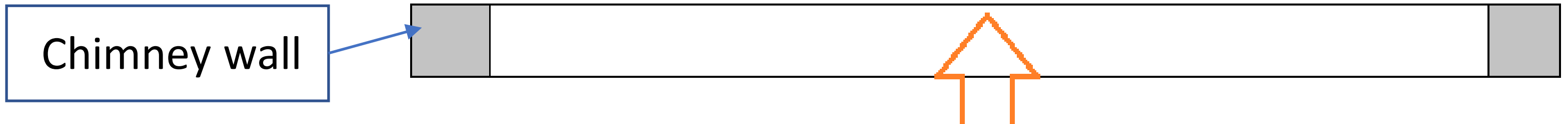
CFD in chimney stack design

Transient analysis:

Temperature profile

at the outlet





3. Future Work: CFD Simulations of Effects of Lining Systems

- Lining systems:
  For example, a stack can be internally covered with borosilicate glass block lining system (BGBLS), commonly known as Pennguard, as supplied by Hadek Protective Systems
- Lining systems are mostly installed for corrosion protection
- CFD simulations can be used to explore other benefits ➔ reduction of ground-level concentration impacts during power plant startups

Air Pollution Models (APM)

- APM simulate ground-level concentration impacts using source, emission, and meteorological inputs
  - E.g., source height, plume exit temperature, wind
- Effective source height = physical height + plume rise
- Plume rise function of ΔT (plume temperature – ambient temperature)
- Plume temperature can be simulated by CFD <u>with and without lining</u> ➔ air quality benefits can be estimated

For example:

- Preliminary simulation of two plume scenarios (flat terrain)
  - Stack height: 50 m
  - Exit velocity: 10 m/s
  - Internal exit diameter: 7 m
  - Exit plume temperature = 60 C (startup) and 90 C (steady state)
- In typical daytime (unstable) and neutral conditions:
  - Maximum ground-level concentration is 50% higher
  - Percentage may be higher in complex terrain
- Plume temperature will be simulated by CFD with and without lining ➔ air quality benefits will be estimated – reduction of ΔH !

Thanks!

Paolo Zannetti, The EnviroComp Institute

zannetti@envirocomp.org

Giuseppe Bucci, SINAPSI Innotec Srl

pino.bucci@sinapsi.net

We acknowledge the ongoing support of

Hadek Protective Systems for our work