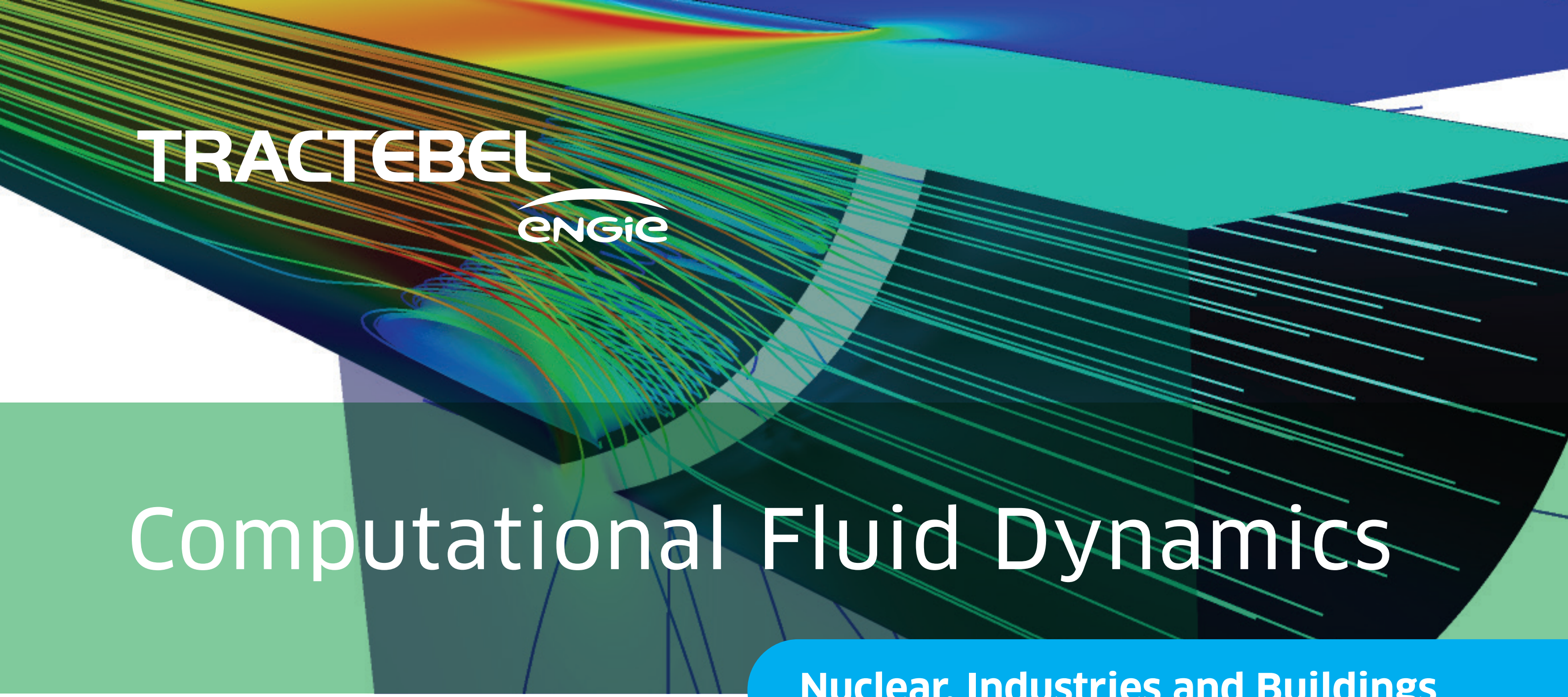


# Computational Fluid Dynamics

## Nuclear, Industries and Buildings

The power of CFD available for nuclear and non-nuclear technical challenges

For more than 150 years, we have provided support to industrial installations around the world. Our high-level engineering and consulting teams give clients valuable support, in every aspect of their projects and business with an uncompromising approach to safety.

We help you design and make decisions based on our advanced knowledge in computational fluid dynamics and heat transfer simulations.

### Introduction

CFD (Computational Fluid Dynamics) is a branch of fluid mechanics. It uses applied mathematics, physics and computational software to visualize how fluids flow (gases or liquids) and to solve heat transfer problems. The technique is very powerful and spans a wide range of industrial and non-industrial applications (see on the reverse side for examples and references).

The domain of application of CFD is very wide. At Tractebel, we specialized and organized our CFD activities around 3 pillars: (1) Design, (2) System Analysis & Diagnostics and (3) Research and Development.

### Tractebel's Added Value

CFD is a modelisation technique. It is important to interpret and validate the obtained results. Over the years, we created an extended validation base for different types of problems. In practice this means that the results obtained by calculations and simulations are compared with experimental test results.

As such the uncertainties can be quantified and the design can be optimized without considering unnecessary conservatisms. This latter aspect helps in optimizing project's lead times and investment costs.

### Coupling of Codes

At Tractebel, we use state-of-the-art CFD computer codes. On top, we have the experience and competence to couple these CFD codes to global thermal-hydraulic codes. This coupling approach allows to bring detailed fluid information to any other computer codes modeling the system global response.

The coupling of codes and our multidisciplinary expertise in house allow to develop an integrated approach when analyzing or designing systems in order to tackle your most complex problems.

### Proven Track Record

We have developed several methodologies in the field of heat transfer, pressure drop and flow mixing calculation for nuclear and non-nuclear applications.

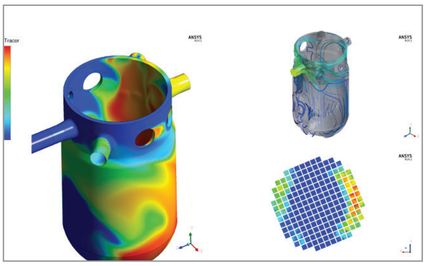
These CFD methodologies provide a fast and reliable solution for multi-physics engineering problems.

5 different projects for which CFD has been successfully used are presented here on the reverse side. These projects are characterized by Multiphysics, 3-dimensionnal fluid and heat transfer phenomena that require accurate modelling for optimum design, root cause analysis or nuclear safety demonstration.

The power of CFD available for nuclear and non-nuclear technical challenges

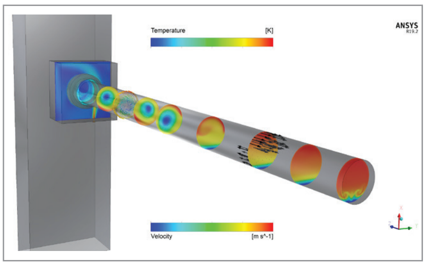
## Projects / References

## Computational Fluid Dynamics



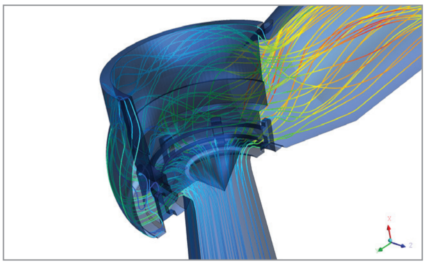

### Boron concentration transient field in a Reactor Pressure Vessel

When one must determine the transient boron concentration field in the Reactor Pressure Vessel in case of heterogeneous dilution i.e. ingress of slug of non-borated water, the only available method is CFD.
Indeed, CFD allows computing the boron field concentration on a very refined spatial scale: the reactor pressure vessel is discretized in millions of cells on which the boron concentration is computed. Such a refinement in the calculated data is necessary to estimate the core reactivity response



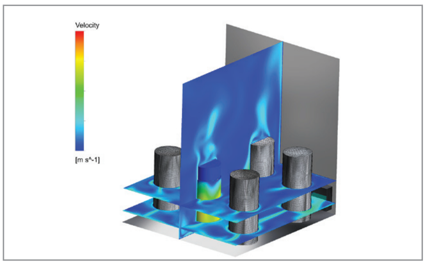

### Thermal fatigue induced by complex flow structures

Dead-end lines are known to be prompt to thermal fatigue as significant temperature differences could exist between hot fluid flowing in main lines and the one cooled by thermal losses.
When one line suffers degradation from thermal fatigue, CFD can isolate the root cause by providing coupled velocity and temperature fields. Complex flow structures such as swirl or natural convection loops can be identified leading to design and safety improvements.
The performed analyses have been supported by experimental measurements conducted by the Von Karman Institute.



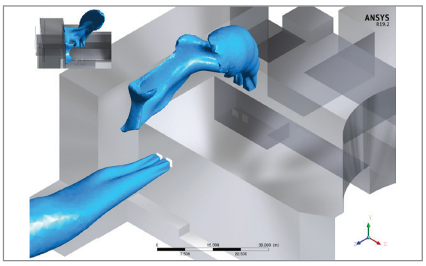
### Compressible flow in a complex geometry – A steam generator safety valve model

Compressible flow i.e. fluid flow with significant density variation is challenging to model as the mass, energy and momentum conservation equations are highly coupled.
CFD has been used to solve this complex equation system to evaluate Steam Generator Safety valve flow capacity for different geometrical configurations. In addition, the opening dynamic has been determined leading to different propositions of design improvements.
This work has been presented at ASME Safety and Relief valve Users Group in 2019.




### Spent Fuel cask storage hall design

The heat transfer process of spent fuel casks in a dry storage facility  includes conduction, convection, radiation as well as natural air circulation.
CFD integrates all the phenomena making possible to model the whole cooling process to end-up with an optimum storage hall design with respect to safety margins, reliability and minimum footprint.
This project has been presented at ENYGF 2019 conference.




### Atmospheric flows: steam discharge on a nuclear site

Atmospheric flows analyses cover all problems related to wind effects  This kind of analyses finds application in environmental engineering for particle dispersion or gas diffusion, wind engineering optimizing the location of wind mills or building engineering.
CFD is the perfect tool fur such application as it allows following particle streamlines and computing wind and concentration fields on large 3-D domains.
In the presented application, it has been successfully applied to the Tihange site to improve workers security in case of steam discharge on site.

**TRACTEBEL**
tractebel-engie.com
nuclear@tractebel-engie.com