The industrial application of CFD

Dr Steve Howell – 1st November 2016

Abercus

Abercus is an independent, privately-owned consultancy specialising in advanced engineering simulation within the energy sector – computational fluid dynamics (CFD), finite element analysis (FEA), the development of bespoke software tools and teaching/training.

Agenda

- Introduction
- Industrial application of CFD
- Lower cost and open source simulation tools
- Verification and validation
- Summary.

Agenda

- Introduction
- Industrial application of CFD
- Lower cost and open source simulation tools
- Verification and validation
- Summary.

Agenda

- Introduction
  - Numerical wind tunnel
  - Discretisation and the CFD process (pre → solve → post)
  - Examples – flow in a pipe, lid-driven cavity
  - Other methods, benefits of CFD
  - General transport equation, convection and diffusion
  - Numerical diffusion
  - Validation.

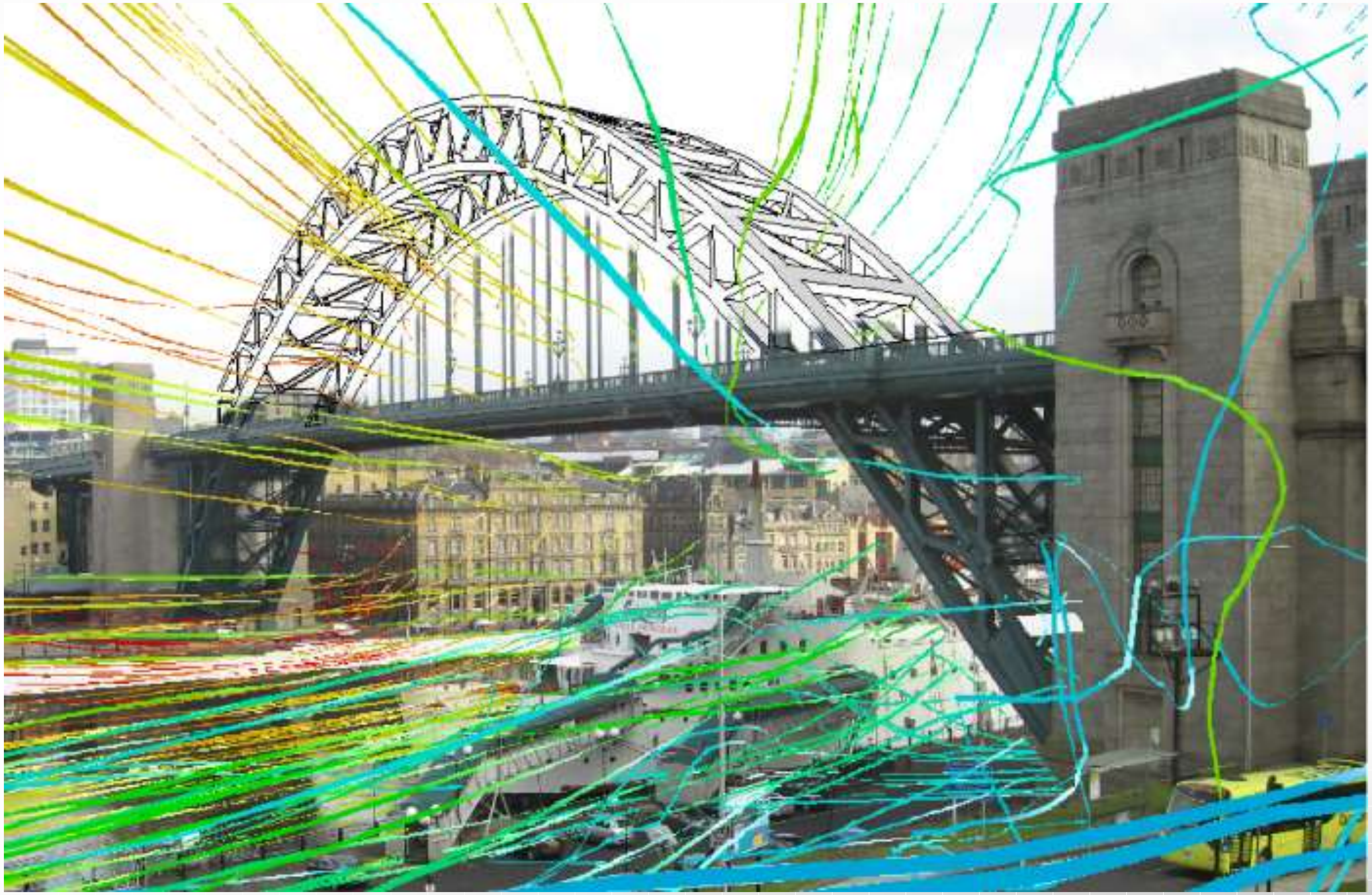
Introduction

- CFD is an acronym for the term computational fluid dynamics

- Computational – using computers to solve a set of equations
- Fluid – (typically) liquid or a gas
- Dynamics – motion

- Computational fluid dynamics is an approach for solving the governing equations of fluid flow using computational methods.

Introduction



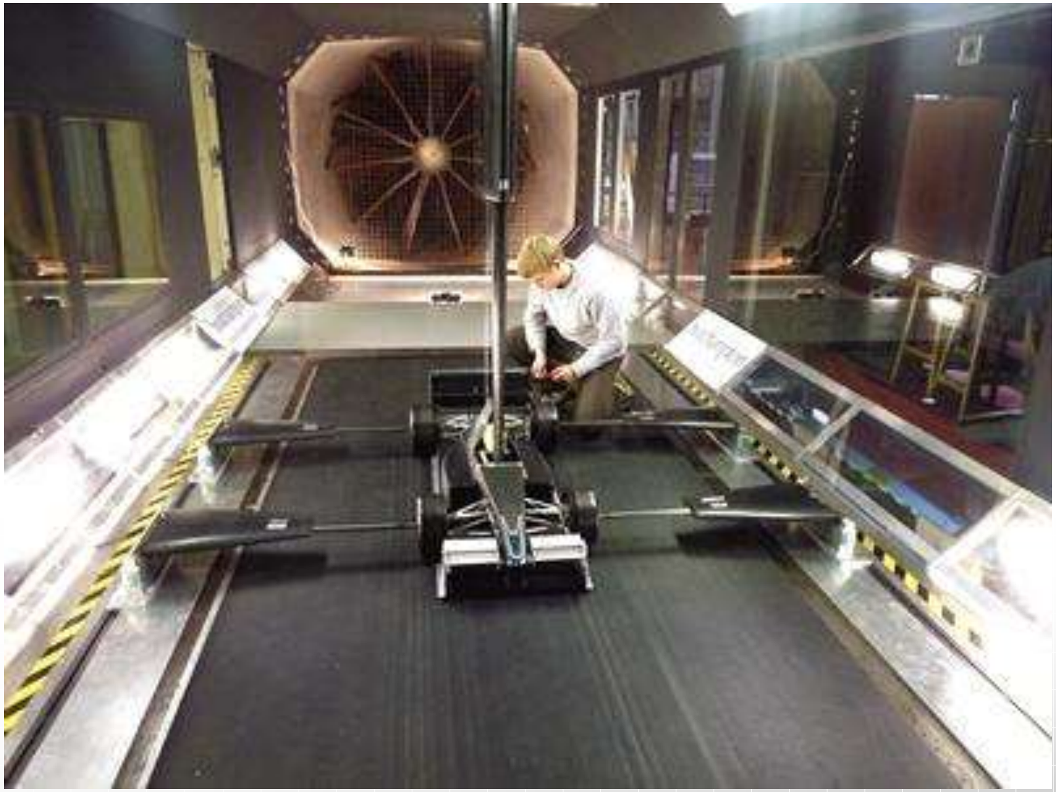
Introduction

- CFD is often described as a numerical wind tunnel or wave tank

RJ Mitchell wind tunnel, University of Southampton:
http://www.southampton.ac.uk/engineering/research/facilities/360/wind_tunnel_r_j_mitchell.page

Introduction

- CFD is often described as a numerical wind tunnel or wave tank



RJ Mitchell wind tunnel, University of Southampton

Introduction

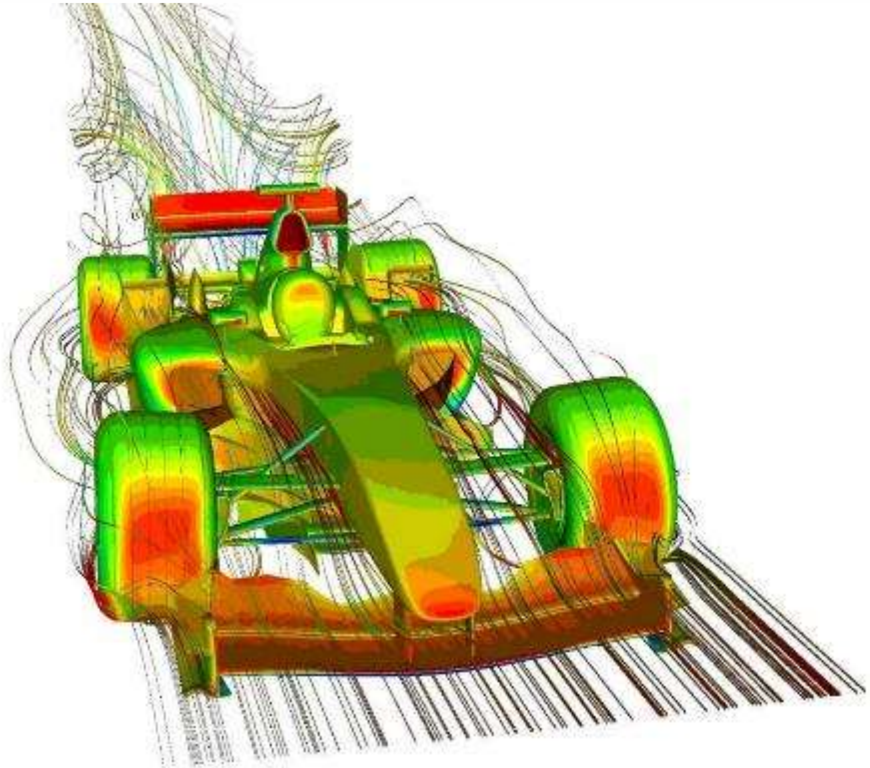
- CFD is often described as a numerical wind tunnel or wave tank

Red Bull Racing


Ferrari

Introduction

- CFD is often described as a numerical wind tunnel or wave tank



CFD simulation by others, reference required

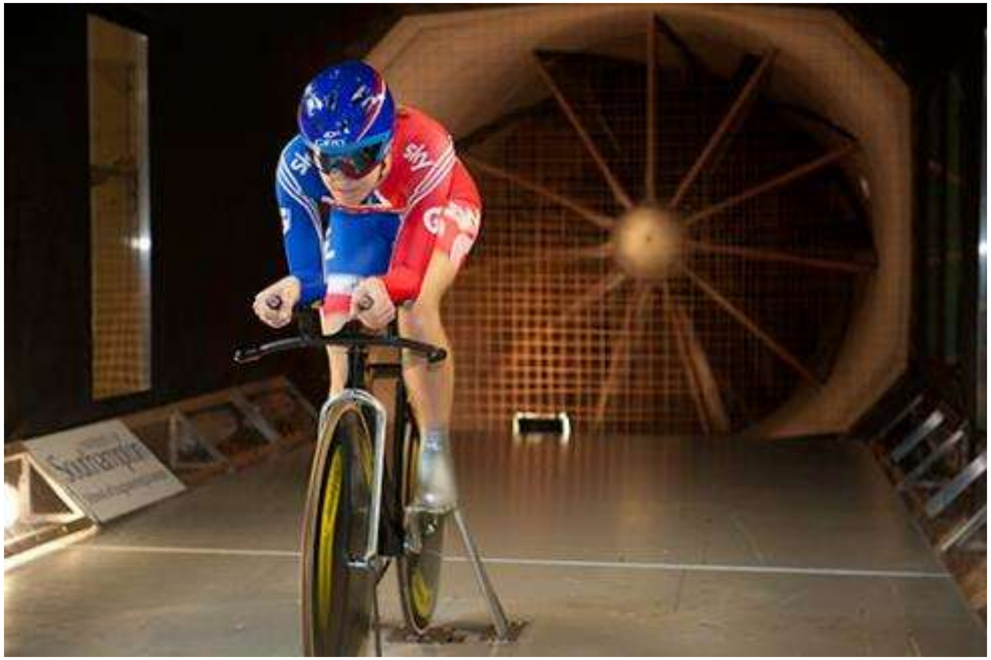
Introduction

- CFD is often described as a numerical wind tunnel or wave tank

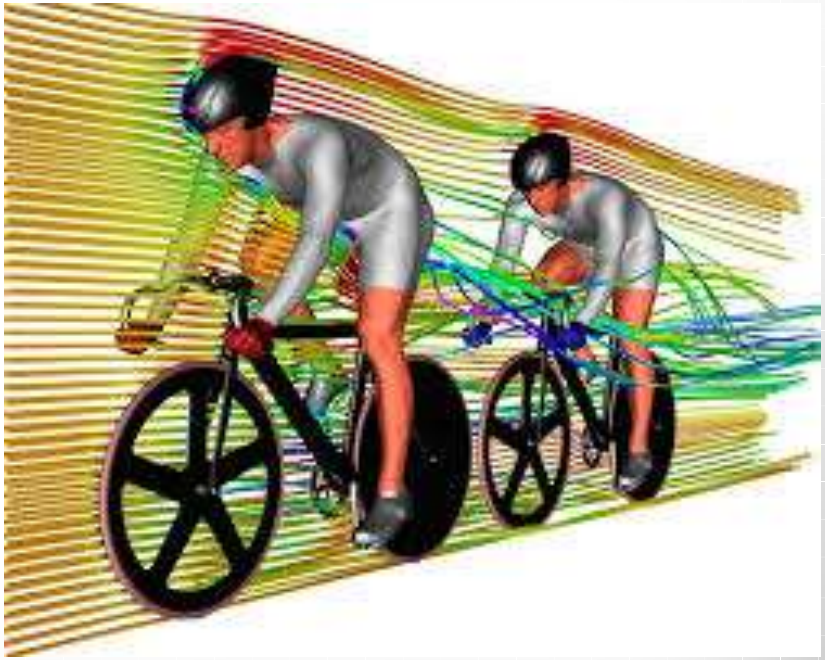
RWDI wind tunnel

Introduction

- CFD is often described as a numerical wind tunnel or wave tank



RJ Mitchell wind tunnel, University of Southampton

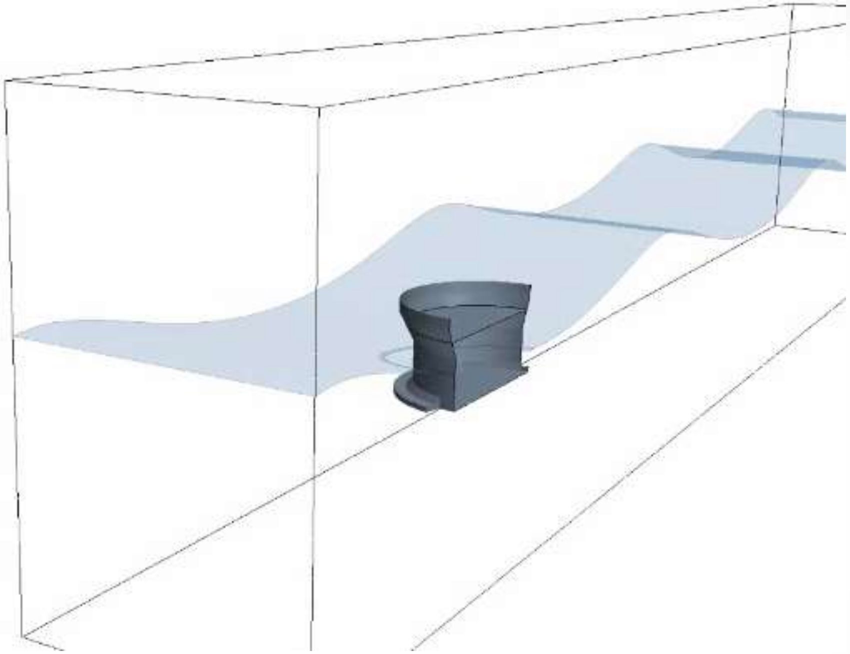


Reference required

Introduction

- CFD is often described as a numerical wind tunnel or wave tank

Wave tank at Marin



CFD by Abercus/Genesis Oil and Gas

Introduction

- CFD is an approach for solving the governing equations of fluid flow using computational methods, but why don't we just solve these equations analytically?
- The governing equations are complex, non-linear partial differential equations

$$\frac{\partial(\rho u_i)}{\partial t}\mathbf{e}_i + \frac{\partial(\rho u_i u_j)}{\partial x_j}\mathbf{e}_i = \mu \frac{\partial^2 u_i}{\partial x_j^2}\mathbf{e}_i + (\rho - \rho_0)g_i\mathbf{e}_i - \frac{\partial \tilde{p}}{\partial x_i}\mathbf{e}_i$$

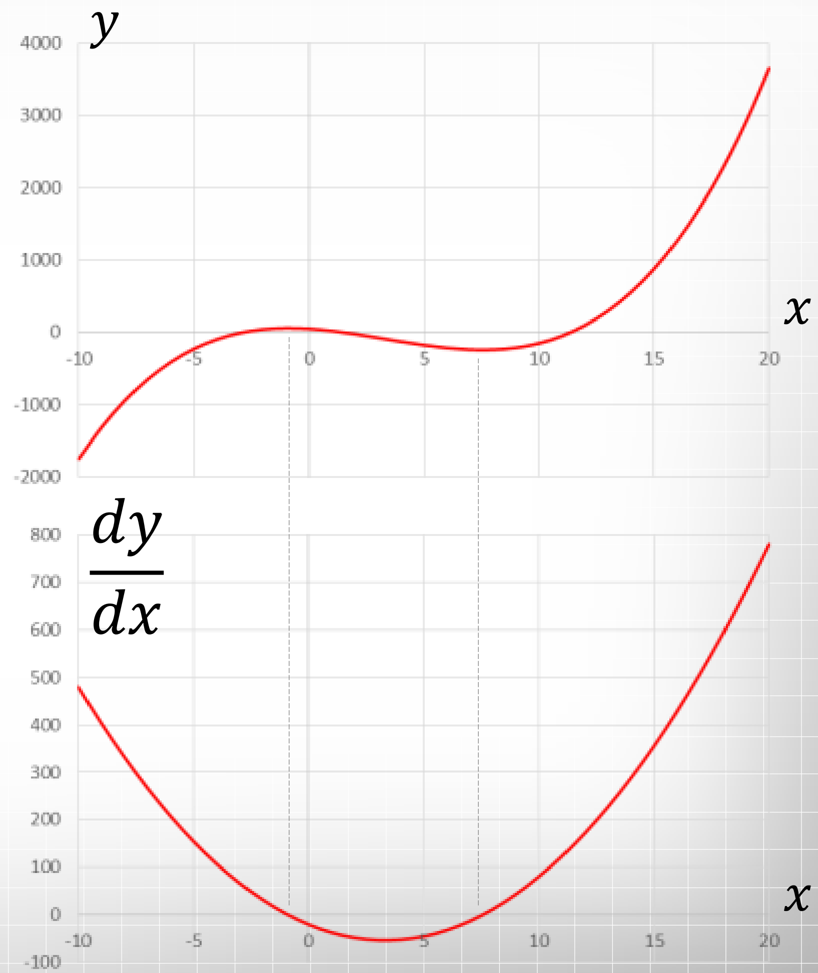
- They have been solved for a few simple geometries, but no general solution is known
- Generally, whenever an analytical solution is not possible, numerical methods offer an alternative approach.

Introduction

- The governing equations contain gradient terms so their solution requires differentiation
- Differentiation may be straight forward if the function is known an linear, for example:

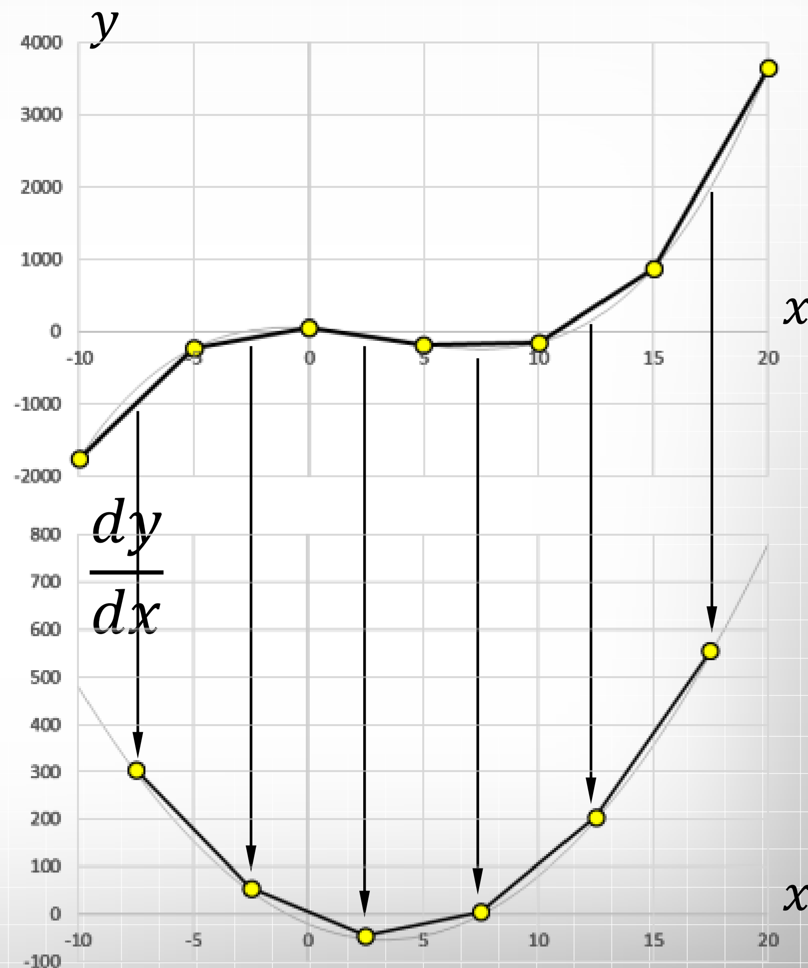
$$y = x^3 - 10x^2 - 20x + 50$$

$$\frac{dy}{dx} = 3x^2 - 20x - 20$$



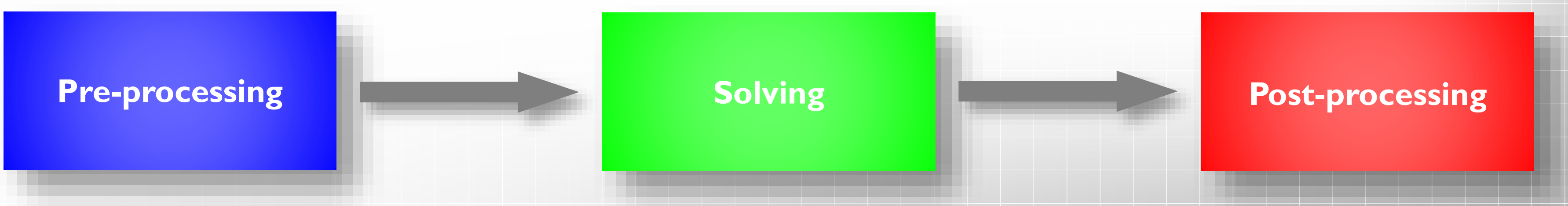
Introduction

- If the function is unknown but the value of the function is known at discrete locations, the gradients can be calculated accordingly
- The original continuous functions are approximated by a system of discrete linear algebraic equations.



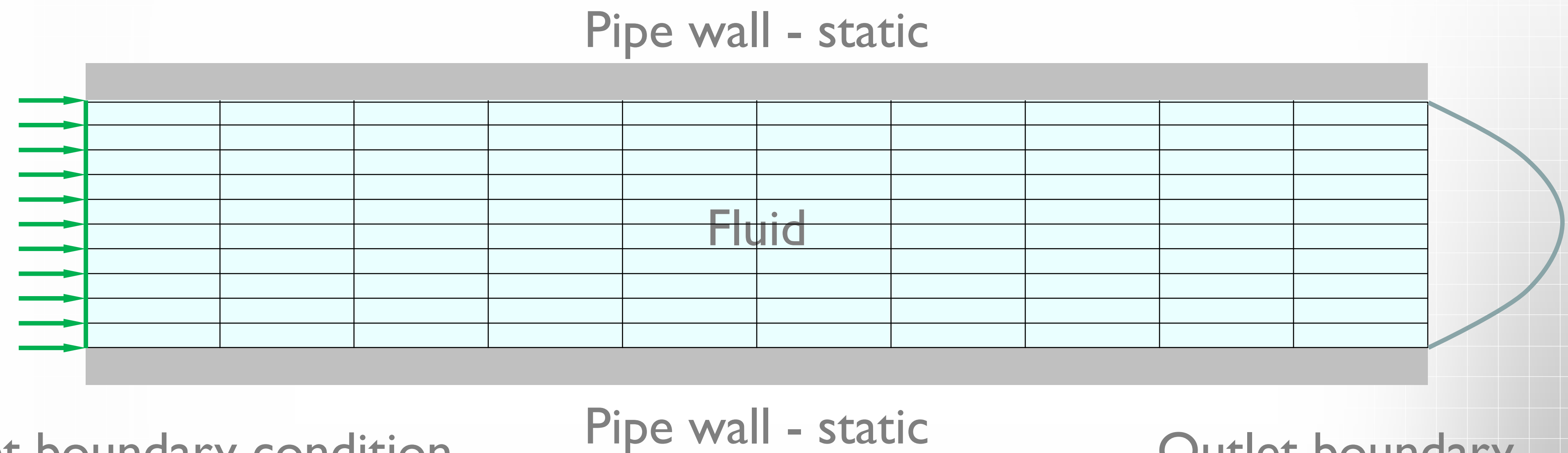
Introduction

- Similarly with CFD, the spatial domain of interest is divided into smaller discrete non-overlapping cells to form a CFD mesh – this is known as discretisation
- The governing equations, which are continuous partial differential equations, are approximated by a system of discrete linear algebraic equations that are solved iteratively
- The numerical information for the solved equation set is interrogated to provide information that is easy to understand.



Introduction

CFD example – flow in a pipe



Inlet boundary condition – uniform velocity

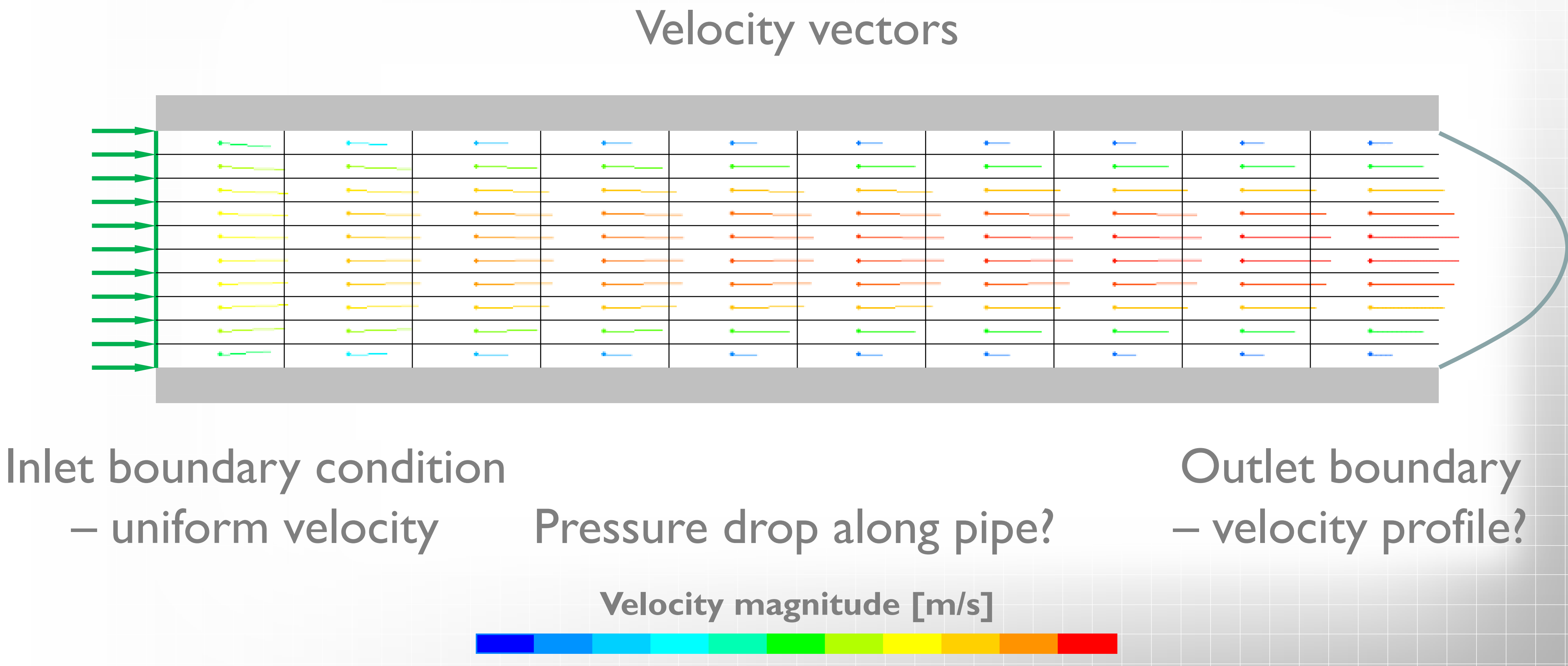
Pressure drop along pipe?

Outlet boundary – velocity profile?

Orthoflo

Introduction

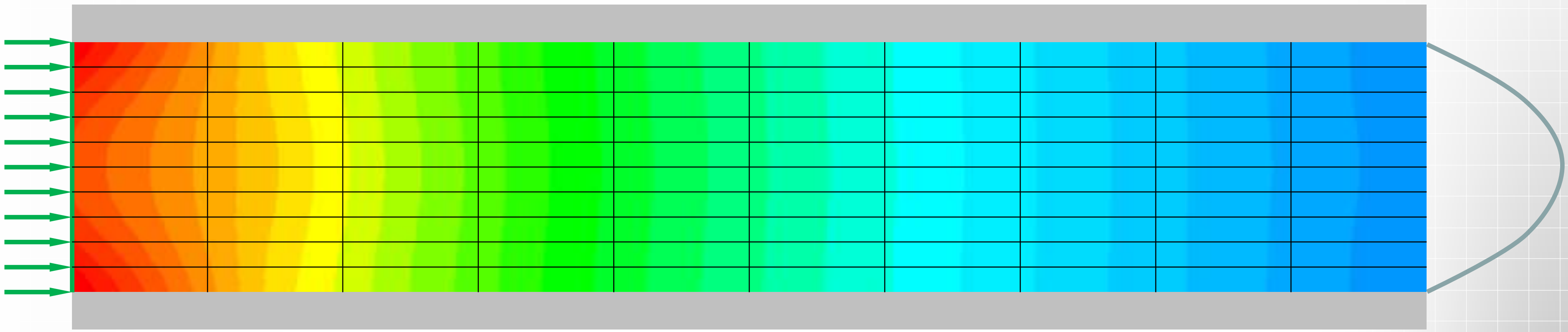
CFD example – flow in a pipe



Introduction

CFD example – flow in a pipe

Pressure contours



Inlet boundary condition – uniform velocity

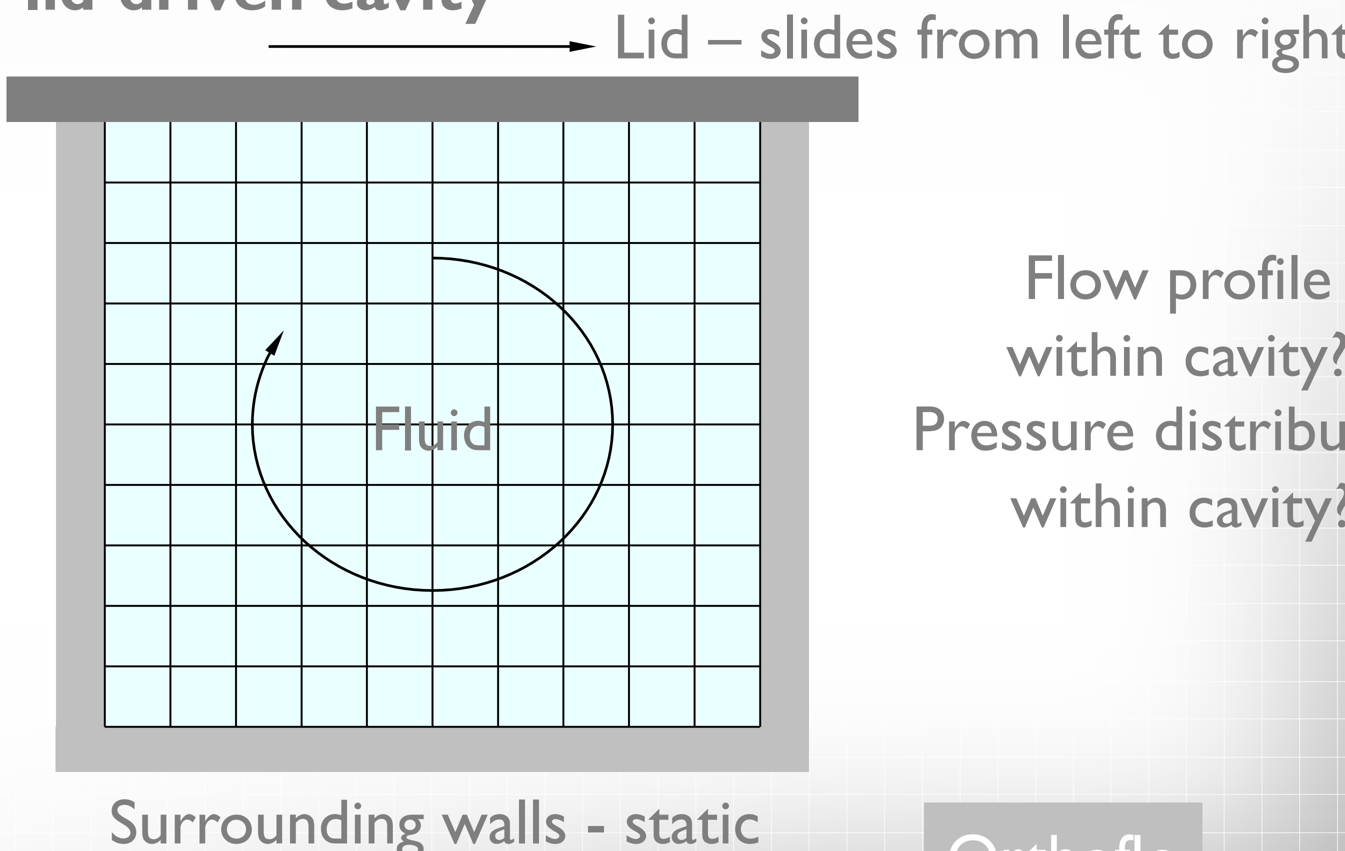
Pressure drop along pipe?

Outlet boundary – velocity profile?

Pressure [Pa]

Introduction

CFD example – lid-driven cavity

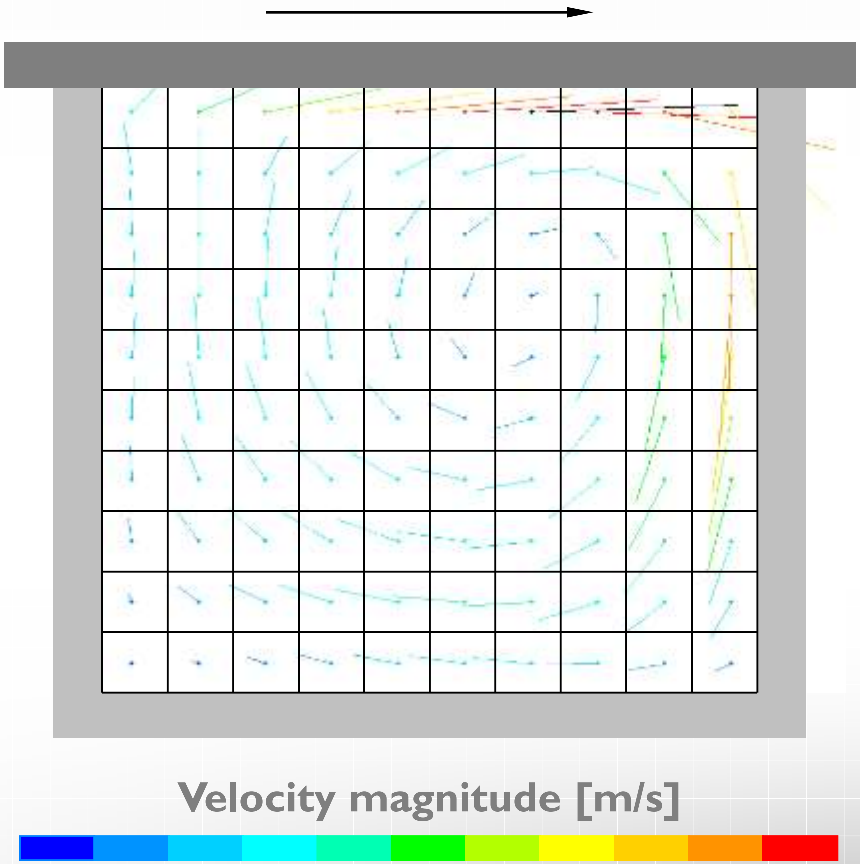


Flow profile within cavity?
Pressure distribution within cavity?

Orthoflo

Introduction

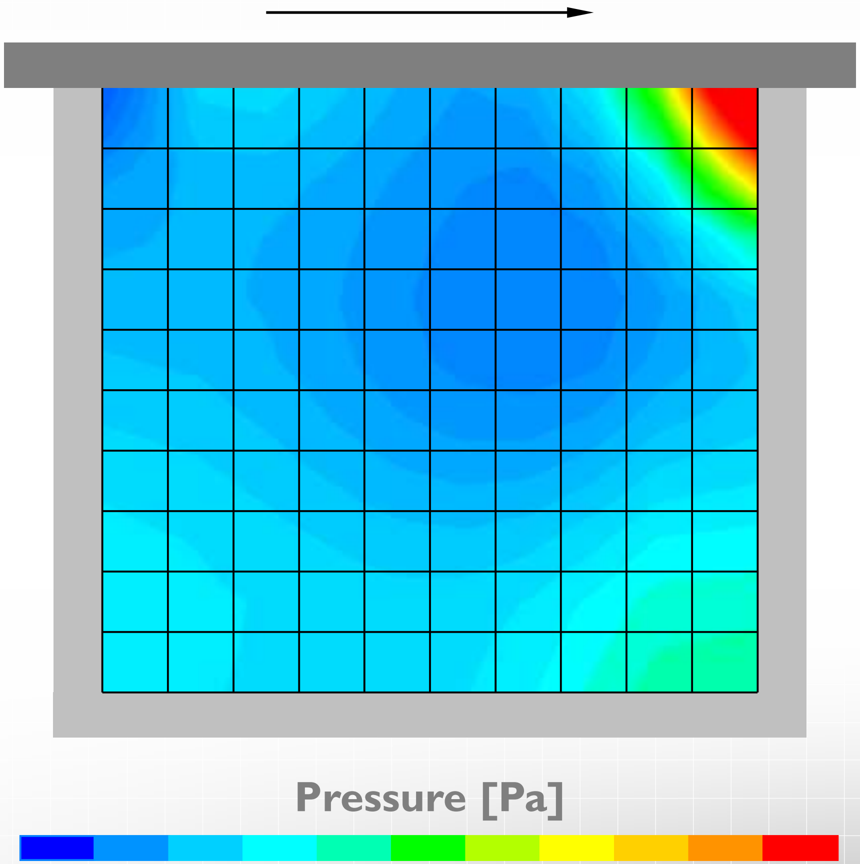
CFD example – lid-driven cavity



Flow profile within cavity?
Pressure distribution within cavity?

Introduction

CFD example – lid-driven cavity



Flow profile within cavity?
Pressure distribution within cavity?

Introduction

Pre-processing → **Solving** → **Post-processing**

Pre-processing
- Geometry preparation
- Discretisation/mesh
- Define fluid properties
- Define boundary conditions
- Define initial conditions

Solving
- Iterative solution

Post-processing
- Numerical information
- Contour plots
- Vector plots

Introduction

- CFD is one approach for solving fluid flow, but there are others:
  - Calculation
  - Experimental correlations
  - Bespoke experiments (small-scale or full-scale)
  - Full-scale experiment
- All of the different methods have their relative strengths and weaknesses, and are appropriate in different applications.

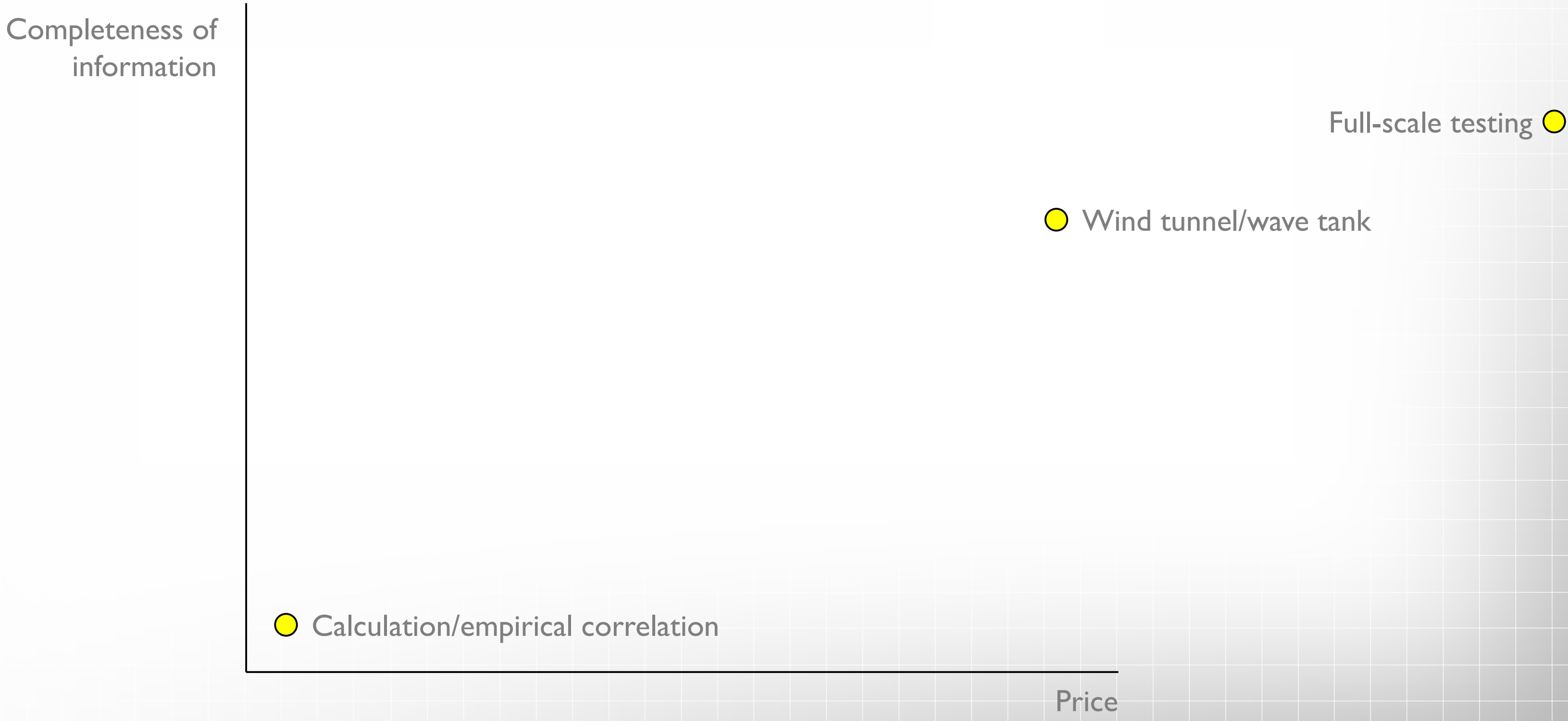
Introduction

- CFD is one approach for solving fluid flow, but there are others:

Approach	Advantages	Disadvantages
Calculation	Quick and easy for simple flow geometries	Is the flow properly captured? May be overly conservative?
Empirical correlation	Quick and easy Experimental validation	Does a valid correlation exist for the flow/geometry of interest? Is the correlation overly conservative?
Small scale experiment	Allows complex geometries to be considered Controlled conditions	Scaling effects – is the flow Reynolds number independent? Information only recorded at prescribed monitor locations
Full-scale experiment	Real world, real geometries No scaling effects	Uncontrollable conditions Information only recorded at prescribed monitor locations

Introduction

- CFD is one approach for solving fluid flow, but there are others:



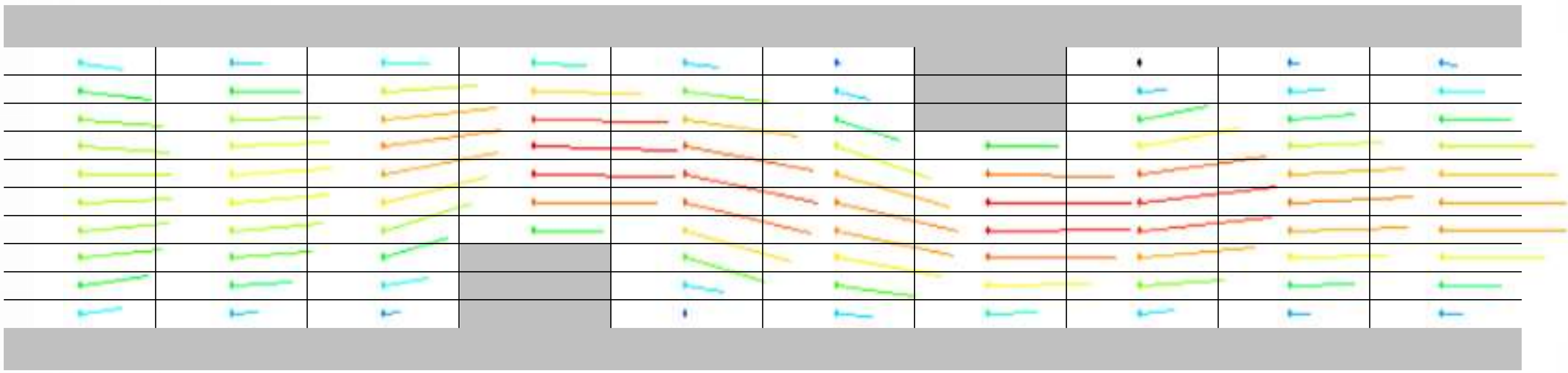
Introduction

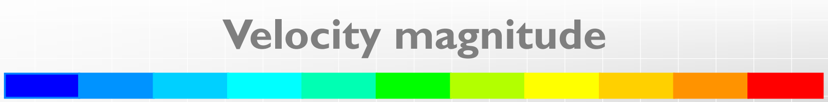
- CFD simulates the flow at full-scale, so there are no issues with scaling effects
- Changes in the CFD model can be quickly incorporated, both in terms of the model geometry and/or the boundary conditions, which allows sensitivities to be considered and optimisation to be undertaken

Orthoflo

Introduction

Changes to geometry and boundary conditions can be quickly investigated.



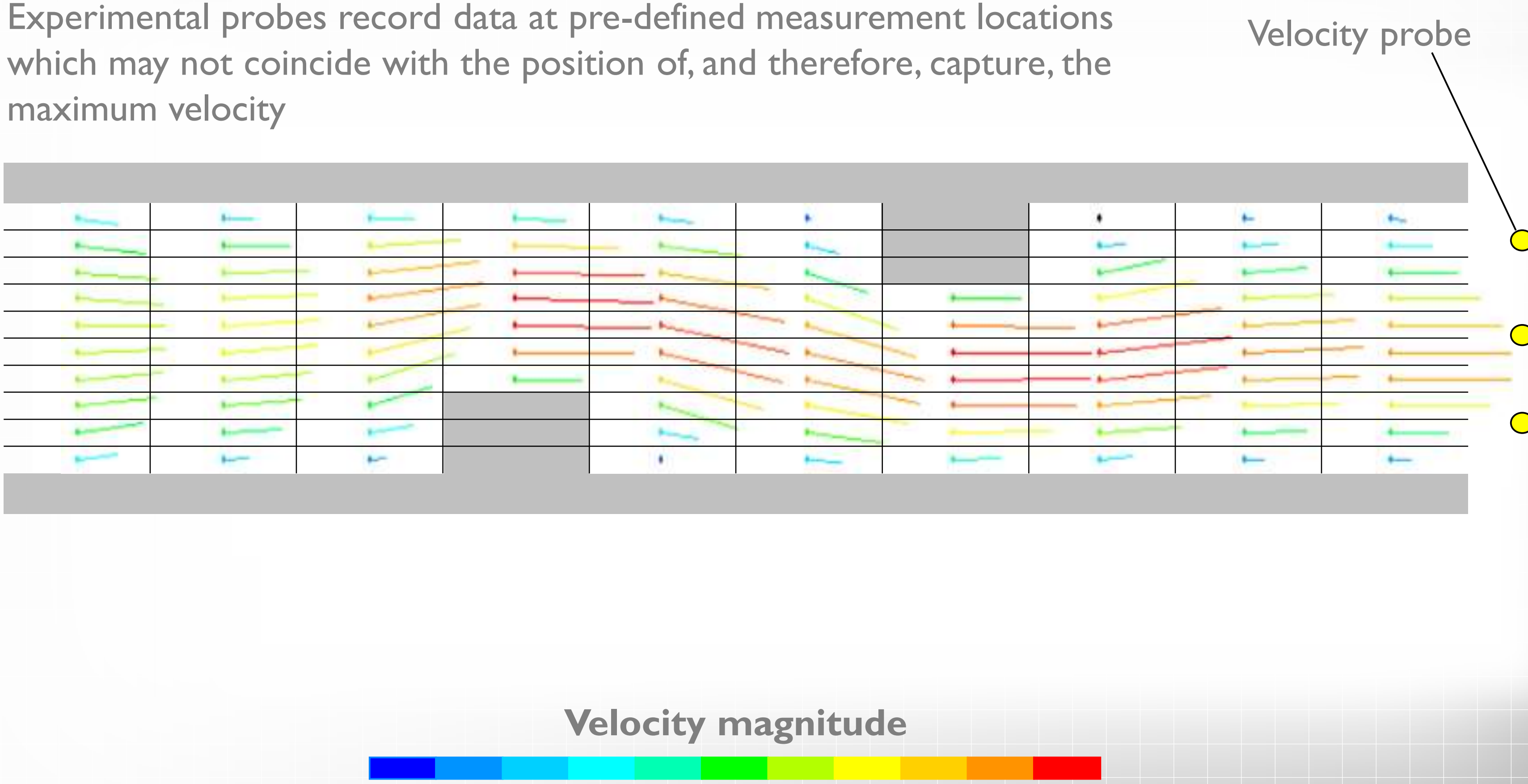


Introduction

- CFD simulates the flow at full-scale, so there are no issues with scaling effects
- Changes in the CFD model can be quickly incorporated, both in terms of the model geometry and/or the boundary conditions, which allows sensitivities to be considered and optimisation to be undertaken
- CFD provides a complete solution throughout the spatial domain, not just at pre-defined probe locations, so the predicted flow behaviour can be interrogated at any point within the domain.

Introduction

Experimental probes record data at pre-defined measurement locations which may not coincide with the position of, and therefore, capture, the maximum velocity



Introduction

- Additional flow physics can be incorporated depending upon the application
  - energy
  - turbulence
  - combustion
  - radiation
  - multiphase
  - particle tracking
  - free surface
  - erosion
  - moving objects
  - fluid-structure interaction

Introduction

Conservation equations (Eulerian framework)

- Mass

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_i)}{\partial x_i} = 0$$

- Linear momentum (in vector notation) – Navier-Stokes eqution

$$\frac{\partial (\rho u_i)}{\partial t}\mathbf{e}_i + \frac{\partial (\rho u_i u_j)}{\partial x_j}\mathbf{e}_i = \mu \frac{\partial^2 u_i}{\partial x_j^2}\mathbf{e}_i + (\rho - \rho_0) g_i \mathbf{e}_i - \frac{\partial \tilde{p}}{\partial x_i}\mathbf{e}_i$$

- Energy

$$\frac{\partial (\rho \cdot c_P T)}{\partial t} + \frac{\partial (\rho u_i \cdot c_P T)}{\partial x_i} = k \frac{\partial^2 T_i}{\partial x_i^2}$$

Rate of change term

Introduction

Conservation equations (Eulerian framework)

- Mass

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_i)}{\partial x_i} = 0$$

- Linear momentum (in vector notation) – Navier-Stokes eqution

$$\frac{\partial (\rho u_i)}{\partial t}\mathbf{e}_i + \frac{\partial (\rho u_i u_j)}{\partial x_j}\mathbf{e}_i = \mu \frac{\partial^2 u_i}{\partial x_j^2}\mathbf{e}_i + (\rho - \rho_0) g_i \mathbf{e}_i - \frac{\partial \tilde{p}}{\partial x_i}\mathbf{e}_i$$

- Energy

$$\frac{\partial (\rho \cdot c_P T)}{\partial t} + \frac{\partial (\rho u_i \cdot c_P T)}{\partial x_i} = k \frac{\partial^2 T_i}{\partial x_i^2}$$

Convection term

Introduction

Conservation equations (Eulerian framework)

- Mass

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_i)}{\partial x_i} = 0$$

- Linear momentum (in vector notation) – Navier-Stokes eqution

$$\frac{\partial (\rho u_i)}{\partial t}\mathbf{e}_i + \frac{\partial (\rho u_i u_j)}{\partial x_j}\mathbf{e}_i = \mu \frac{\partial^2 u_i}{\partial x_j^2}\mathbf{e}_i + (\rho - \rho_0) g_i \mathbf{e}_i - \frac{\partial \tilde{p}}{\partial x_i}\mathbf{e}_i$$

- Energy

$$\frac{\partial (\rho \cdot c_P T)}{\partial t} + \frac{\partial (\rho u_i \cdot c_P T)}{\partial x_i} = k \frac{\partial^2 T_i}{\partial x_i^2}$$

Diffusion term

Introduction

Conservation equations (Eulerian framework)

- Mass

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u_i)}{\partial x_i} = 0 \quad +0$$

- Linear momentum (in vector notation) – Navier-Stokes eqution

$$\frac{\partial(\rho u_i)}{\partial t}\mathbf{e}_i + \frac{\partial(\rho u_i u_j)}{\partial x_j}\mathbf{e}_i = \mu \frac{\partial^2 u_i}{\partial x_j^2}\mathbf{e}_i + (\rho - \rho_0) g_i \mathbf{e}_i - \frac{\partial \tilde{p}}{\partial x_i}\mathbf{e}_i$$

- Energy

$$\frac{\partial(\rho \cdot c_P T)}{\partial t} + \frac{\partial(\rho u_i \cdot c_P T)}{\partial x_i} = k \frac{\partial^2 T_i}{\partial x_i^2} \quad +0$$

Source term(s)

Introduction

Conservation equations (Eulerian framework)

– Mass

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u_i)}{\partial x_i} = 0 \qquad +0$$

– Linear momentum (in vector notation) – Navier-Stokes eqution

$$\frac{\partial(\rho u_i)}{\partial t}\mathbf{e}_i + \frac{\partial(\rho u_i u_j)}{\partial x_j}\mathbf{e}_i = \mu \frac{\partial^2 u_i}{\partial x_j^2}\mathbf{e}_i + (\rho - \rho_0) g_i \mathbf{e}_i - \frac{\partial \tilde{p}}{\partial x_i}\mathbf{e}_i$$

– Energy

$$\frac{\partial(\rho \cdot c_P T)}{\partial t} + \frac{\partial(\rho u_i \cdot c_P T)}{\partial x_i} = k \frac{\partial^2 T_i}{\partial x_i^2} \qquad +0$$

– General form of the convection-diffusion equation

$$\frac{\partial(\rho \phi)}{\partial t} + \frac{\partial(\rho u_i \phi)}{\partial x_i} = \Gamma \frac{\partial^2 \phi}{\partial x_i^2} + S_\phi$$

Introduction

Vorticity

- Vorticity ω_i describes the rotation of the flow behaviour and is defined as the curl of the velocity field:

$$\omega_i = \varepsilon_{ijk} \frac{\partial}{\partial x_j} u_k$$

- The vorticity equation is derived from the Navier-Stokes equation:

$$\frac{\partial(\rho_0 \omega_i)}{\partial t}\mathbf{e}_i + \frac{\partial(\rho_0 u_j \omega_i)}{\partial x_j}\mathbf{e}_i = \mu\left(\frac{\partial^2 \omega_i}{\partial x_j^2}\right)\mathbf{e}_i + \frac{\partial(\rho_0 \omega_j u_i)}{\partial x_j}\mathbf{e}_i$$

Introduction

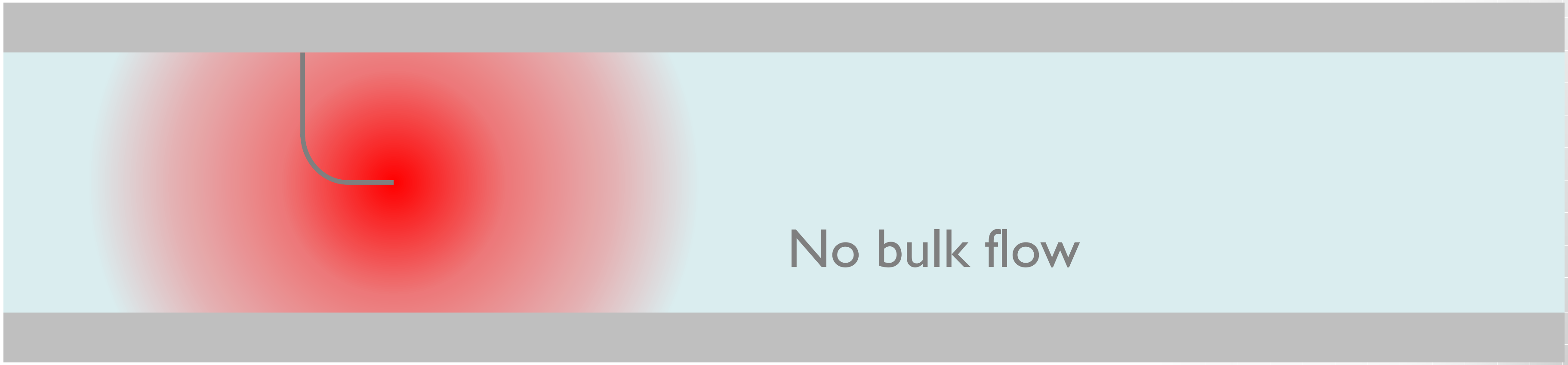
General form of the convection-diffusion equation

$$\frac{\partial(\rho\phi)}{\partial t}+\frac{\partial(\rho u_i\phi)}{\partial x_i}=\Gamma\frac{\partial^2\phi}{\partial x_i^2}+s_\phi$$

Equation	ϕ	Γ	s_ϕ
Conservation of mass	1	0	0
Conservation of *x*-momentum	u	μ	$-\frac{\partial p}{\partial x}$
Conservation of *y*-momentum	v	μ	$-\frac{\partial p}{\partial y}$
Conservation of *z*-momentum	w	μ	$-\frac{\partial p}{\partial z}+(\rho-\rho_0)g$
Vorticity	ω_i	μ	$\frac{\partial(\rho_0\omega_j u_i)}{\partial x_j}$
Energy	T	$\frac{k}{c_P}$	0

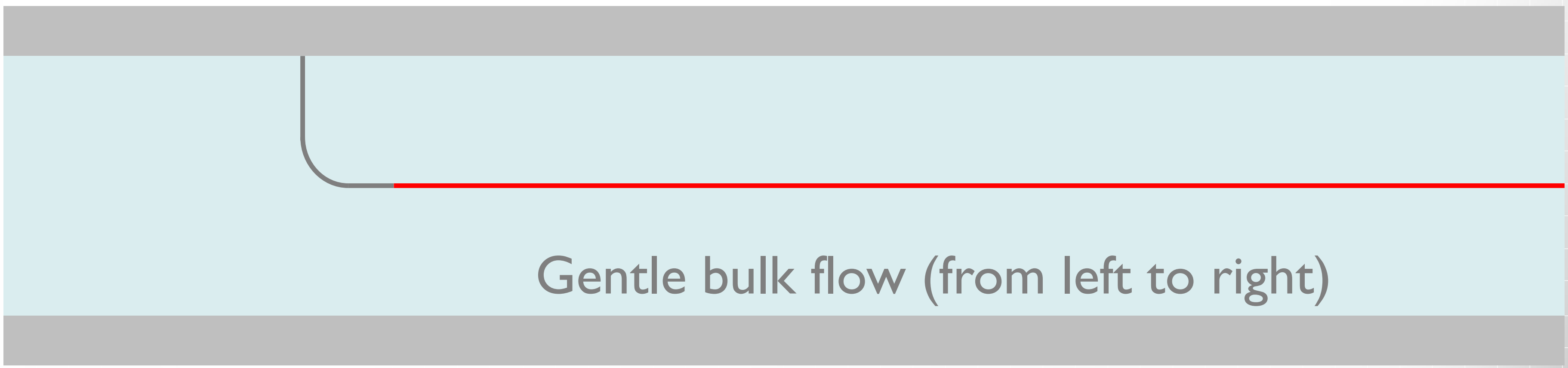
Introduction

- Diffusion – the mixing of fluid in all directions due to random fluctuations at the molecular level



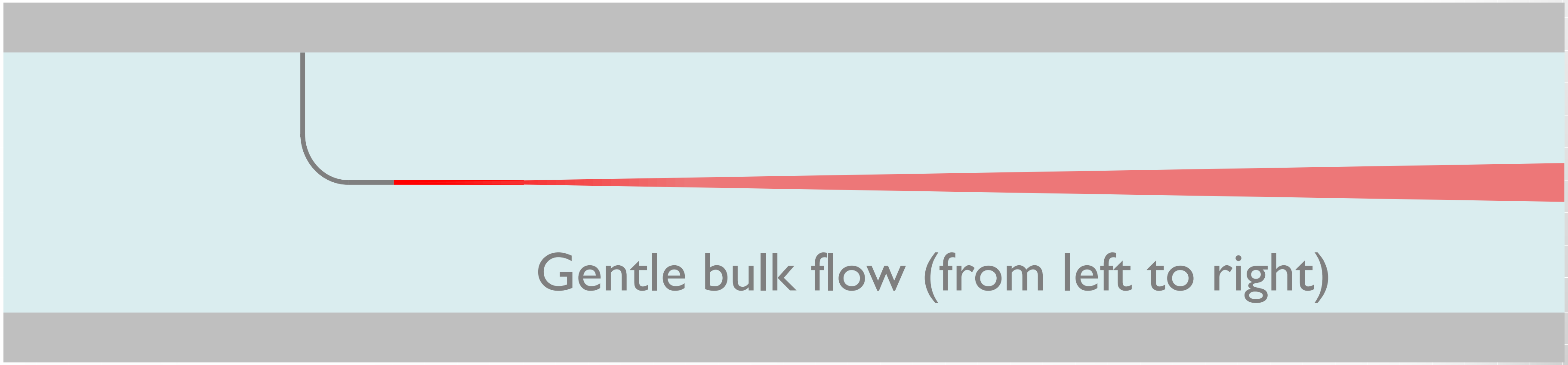
Introduction

- Convection – the collective movement of fluid due to the bulk motion of the fluid



Introduction

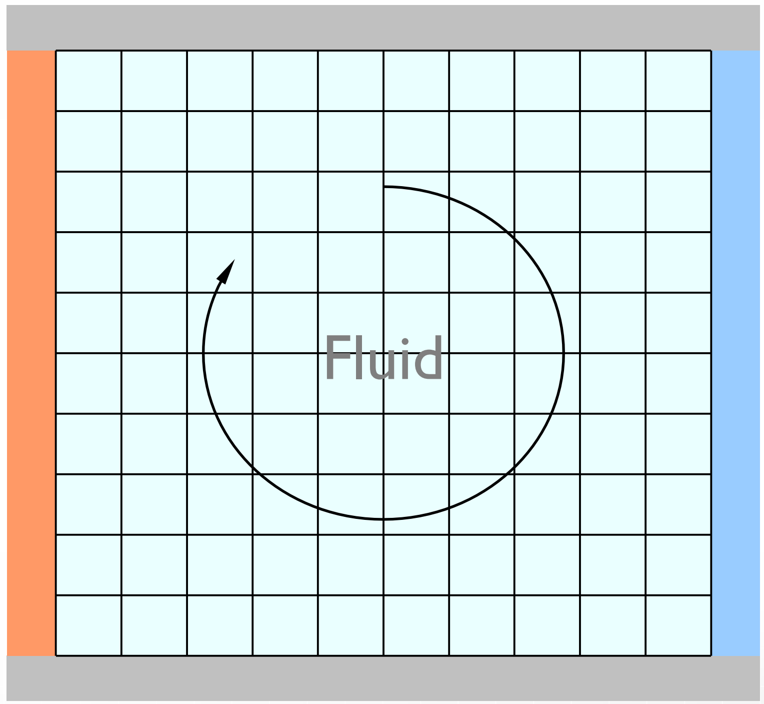
- Typically both convection and diffusion will be present



Introduction

CFD example – flow in a differentially heated cavity

Hot wall



Cold wall

Surrounding walls - static

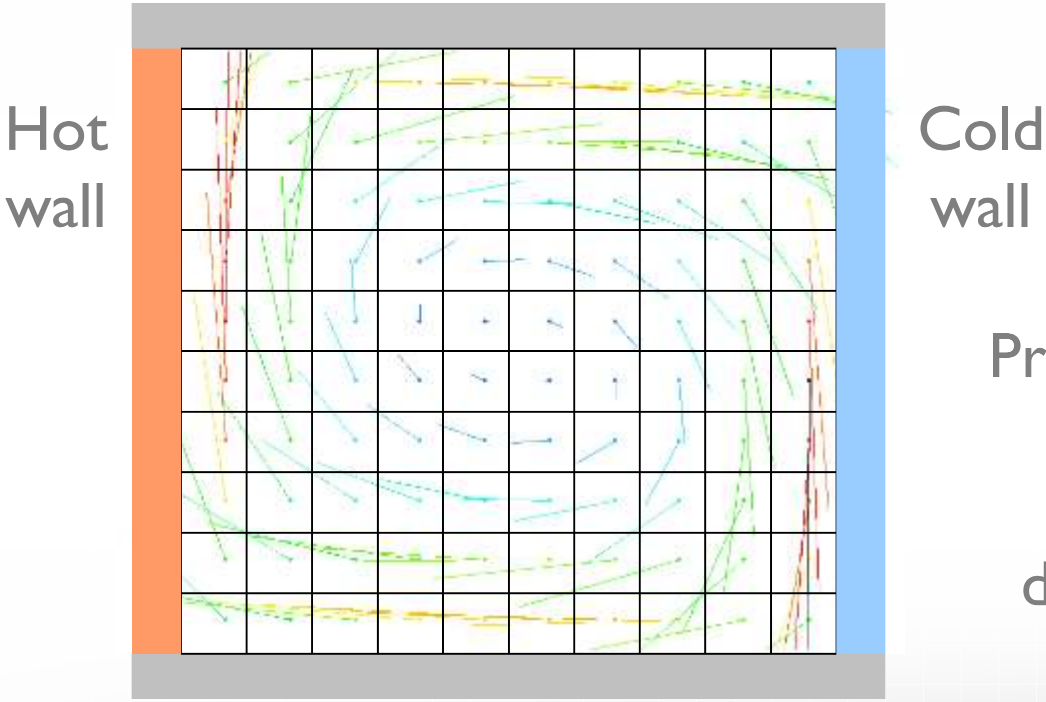
Flow profile within cavity?
Pressure distribution within cavity?
Temperature distribution within cavity?

Orthoflo

Abercus Riverside House Riverside Drive Aberdeen AB11 7LH www.abercus.com

Introduction

CFD example – flow in a differentially heated cavity



Velocity vectors

Flow profile within cavity?

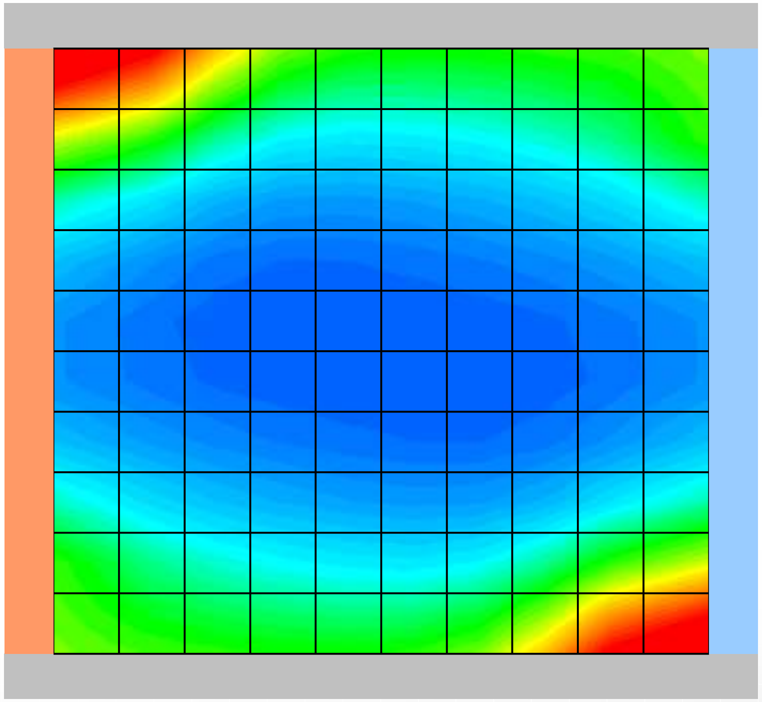
Pressure distribution within cavity?

Temperature distribution within cavity?

Introduction

CFD example – flow in a differentially heated cavity

Hot wall



Cold wall

Flow profile within cavity?

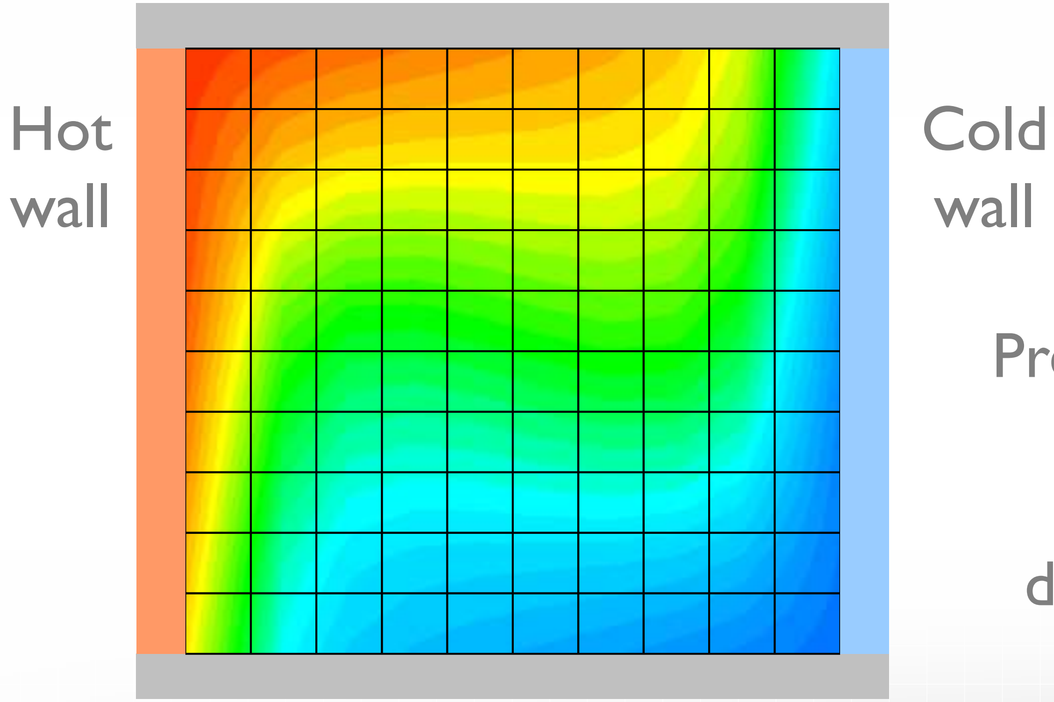
Pressure distribution within cavity?

Temperature distribution within cavity?

Pressure contours

Introduction

CFD example – flow in a differentially heated cavity



Temperature contours

Flow profile within cavity?

Pressure distribution within cavity?

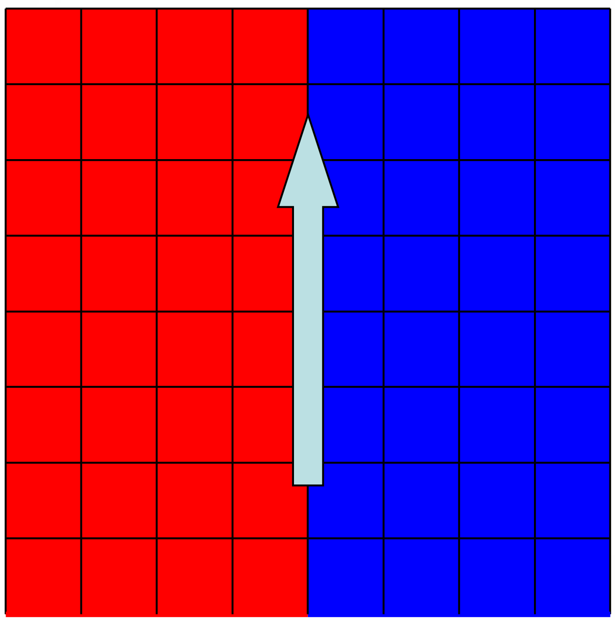
Temperature distribution within cavity?

Introduction

- However…
- CFD approach is restricted by computing resource available (although this is becoming less of a constraint)

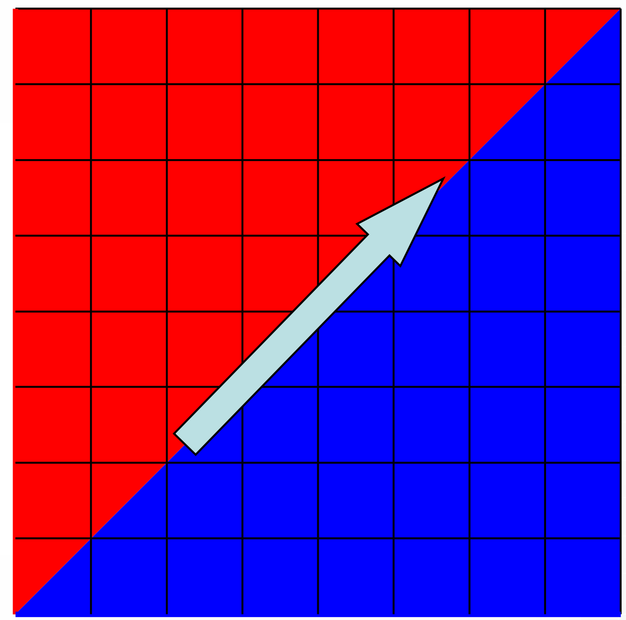
Introduction

Numerical diffusion



$\phi = 1$ $\phi = 0$

$\phi = 1$

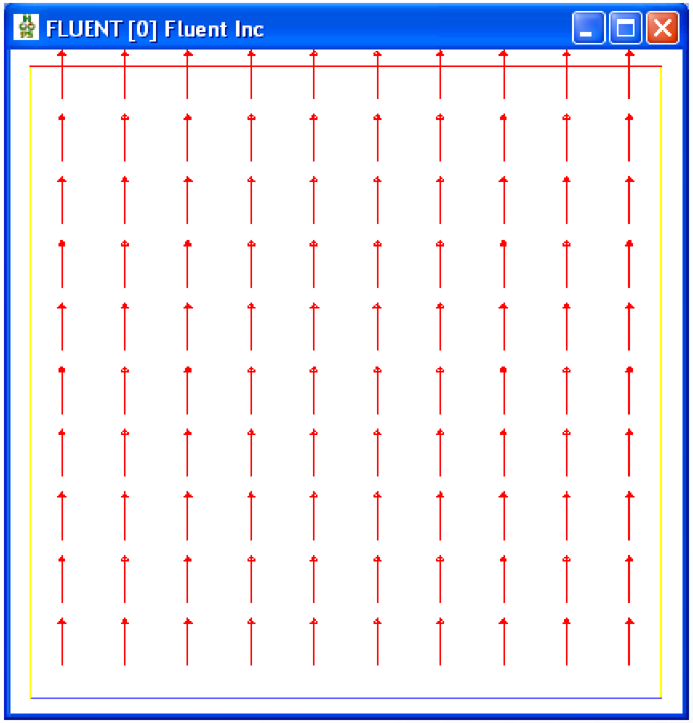


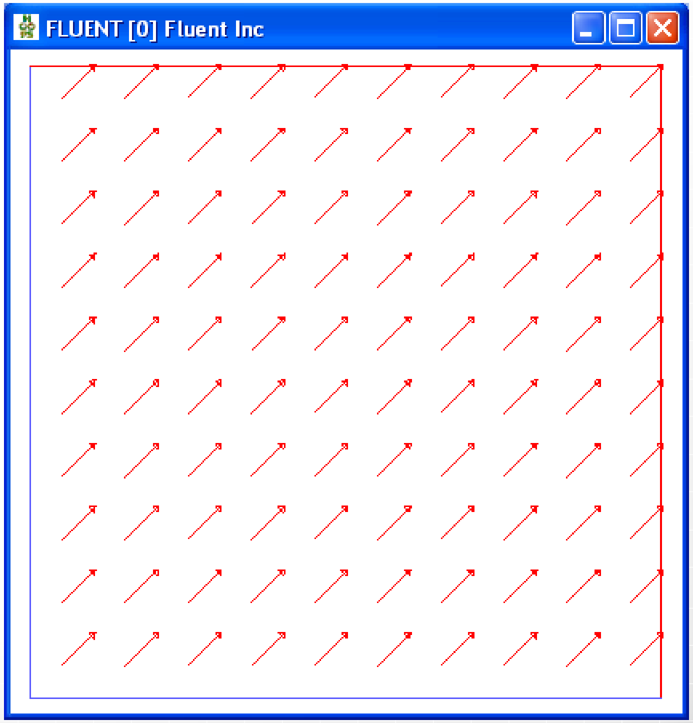
$\phi = 0$

Abercus Riverside House Riverside Drive Aberdeen AB11 7LH www.abercus.com

Introduction

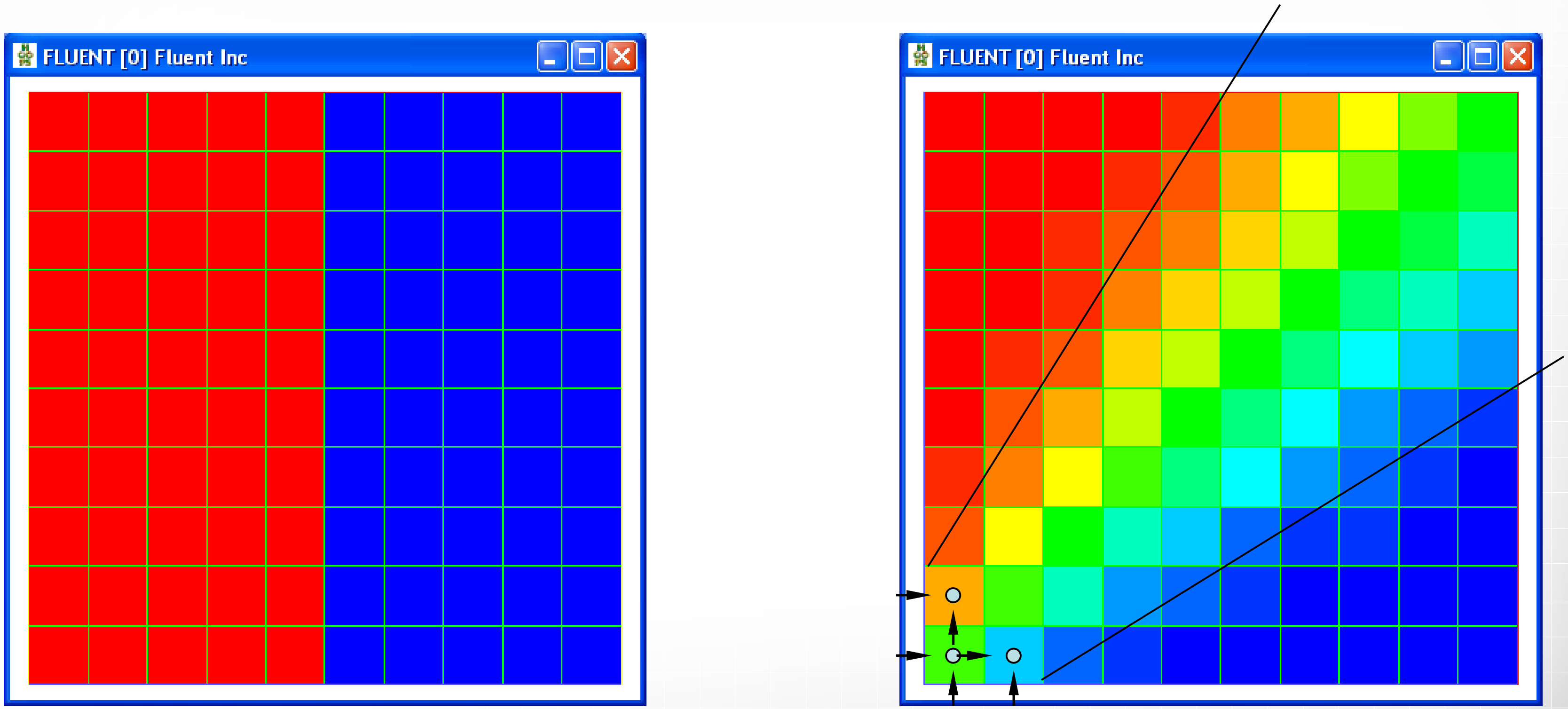
Numerical diffusion





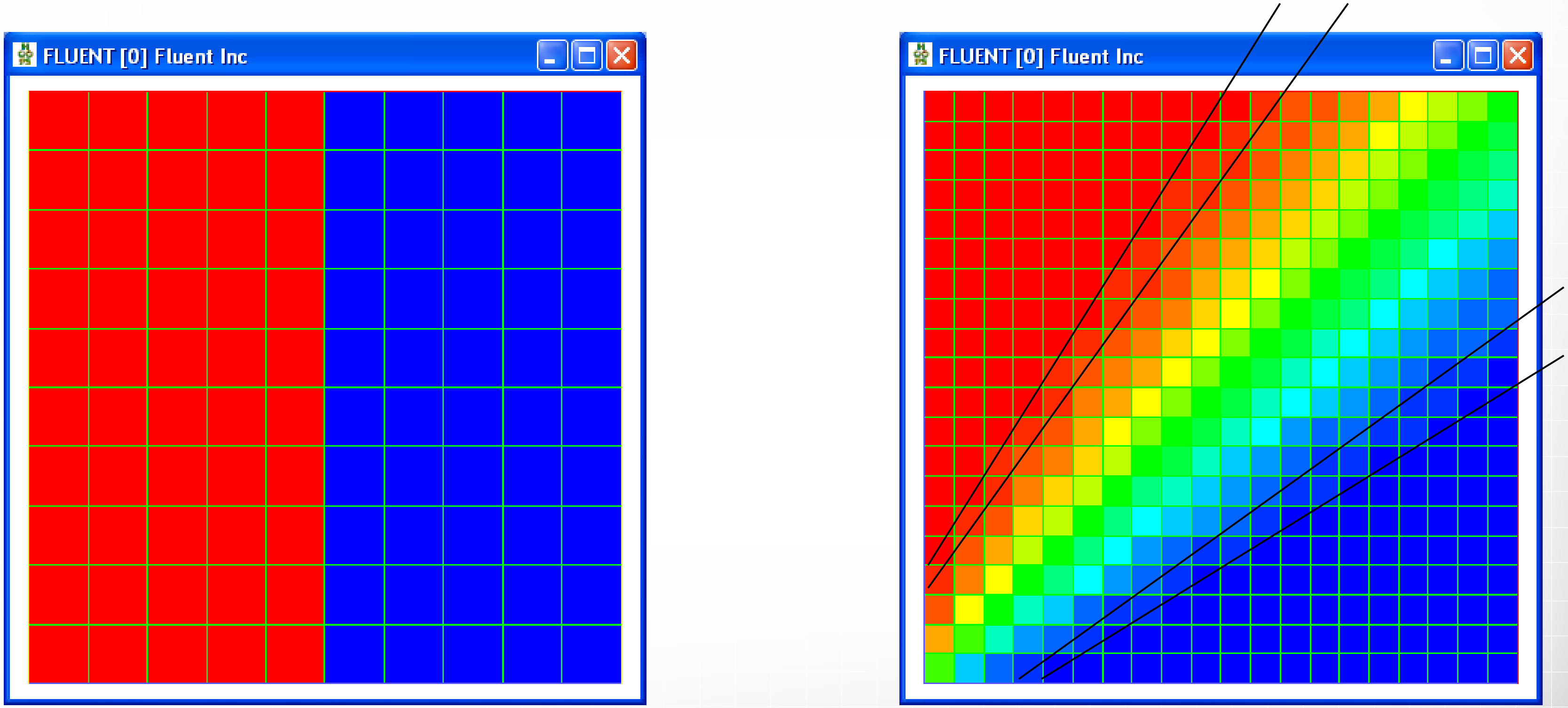
Introduction

Numerical diffusion



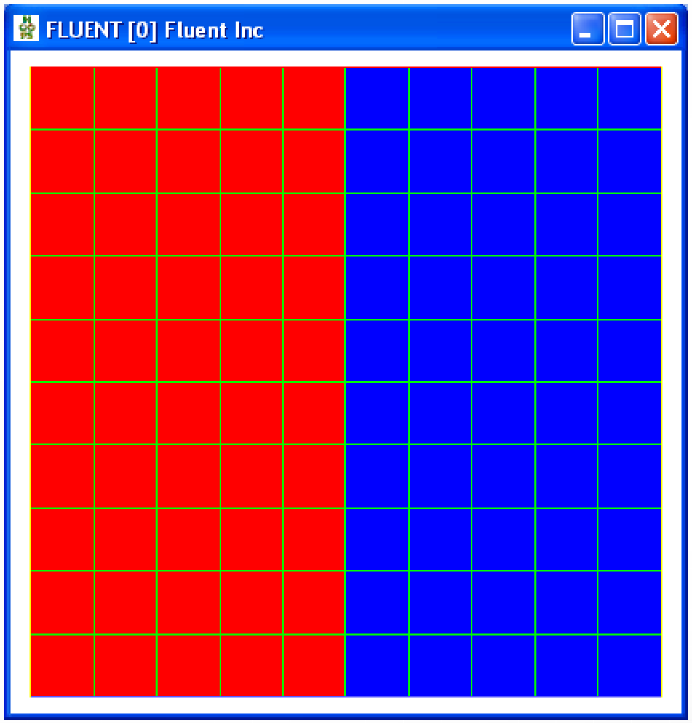
Introduction

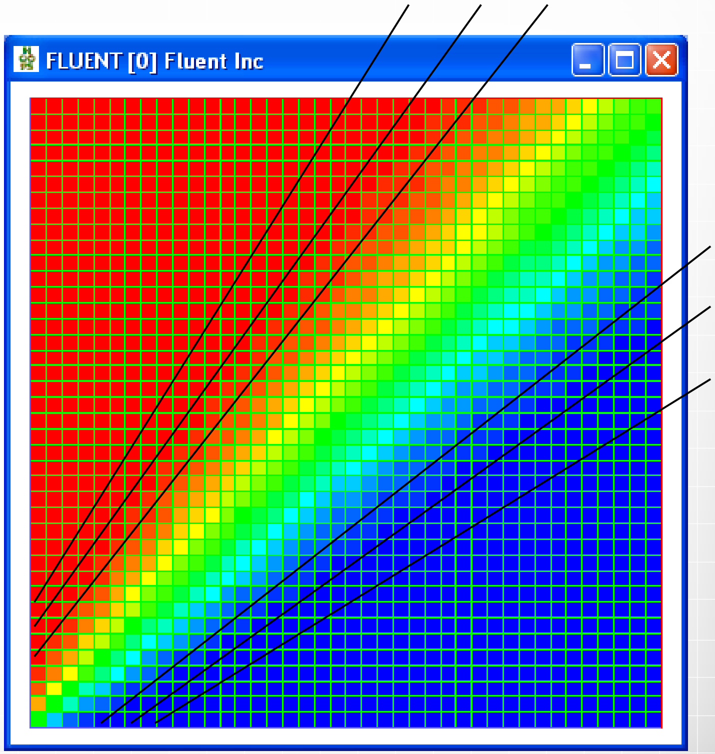
Numerical diffusion



Introduction

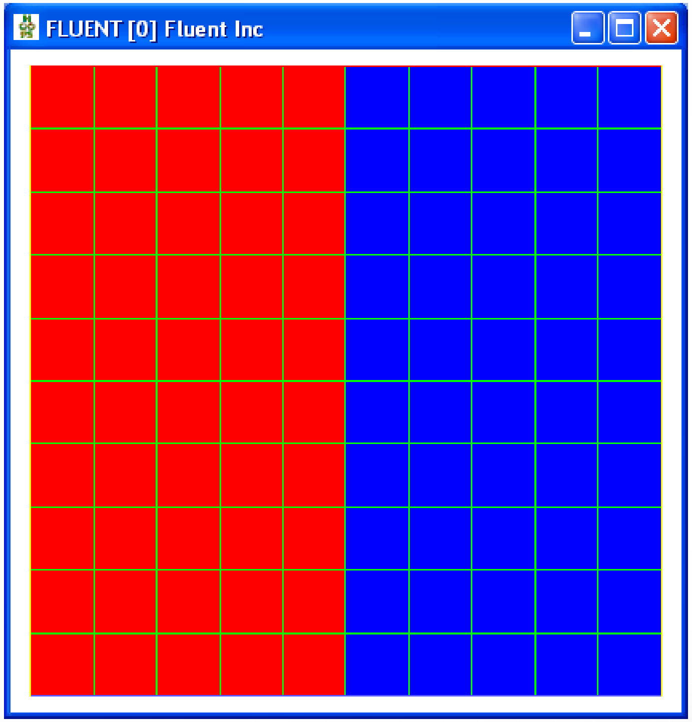
Numerical diffusion

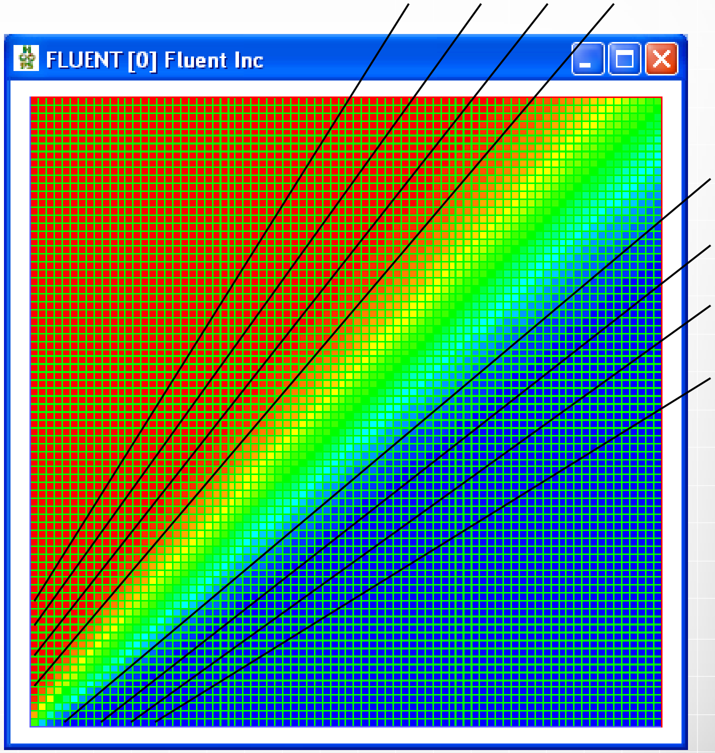




Introduction

Numerical diffusion

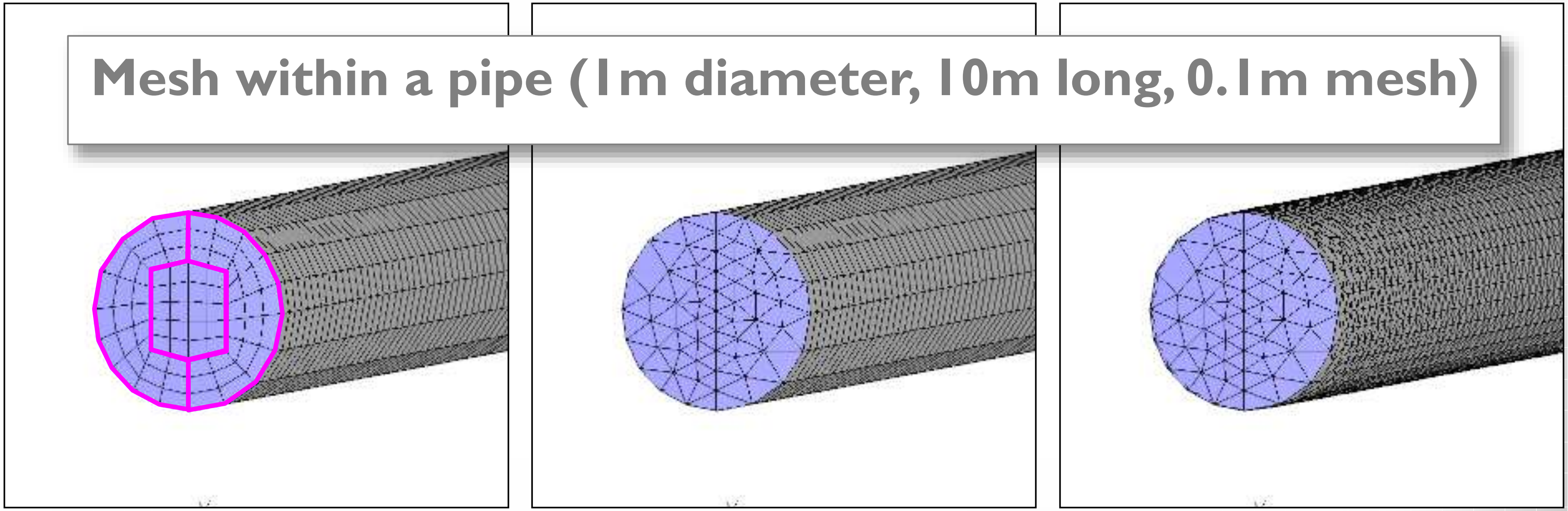




Introduction

Numerical diffusion

Mesh within a pipe (1m diameter, 10m long, 0.1m mesh)

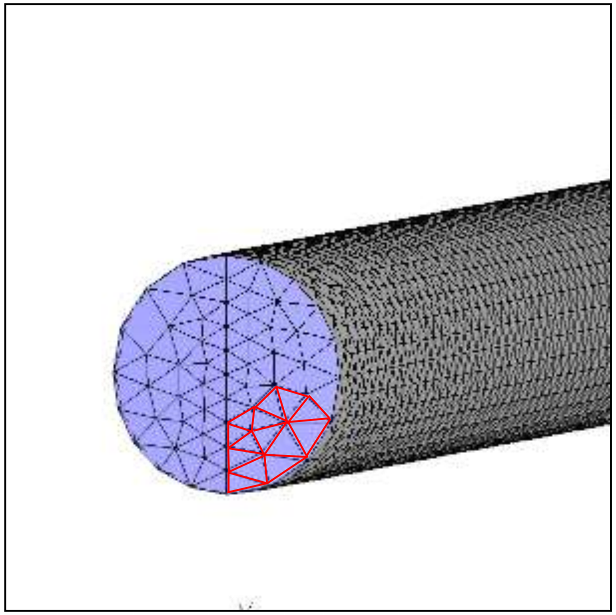


Block structured	Cooper prismatic grid	Unstructured grid
Cell count = 7400	Cell count = 9800	Cell count = 29146
Not as simple to mesh	More straight-forward to mesh	Simplest to mesh

Abercus Riverside House Riverside Drive Aberdeen AB11 7LH www.abercus.com

Introduction

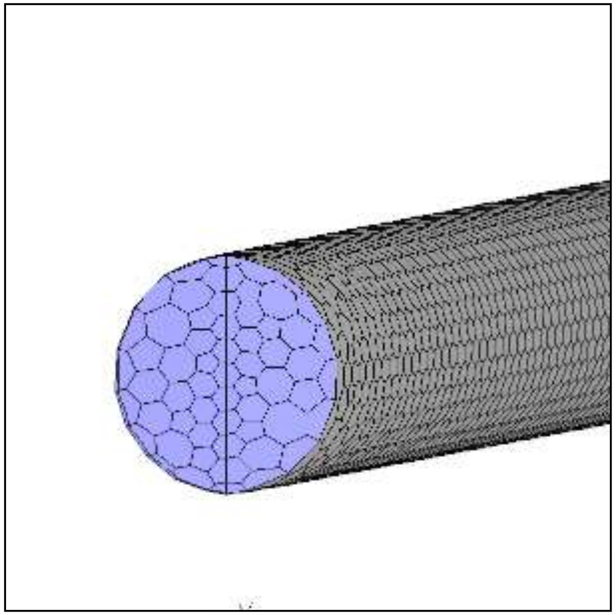
Numerical diffusion



Unstructured grid

Cell count = 29146

Simplest to mesh



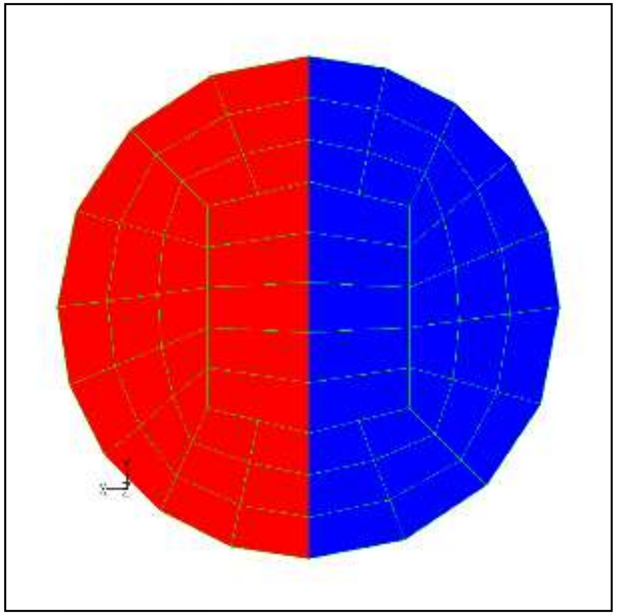
Polyhedral grid

Cell count = 8116

Automatically derived from unstructured mesh

Introduction

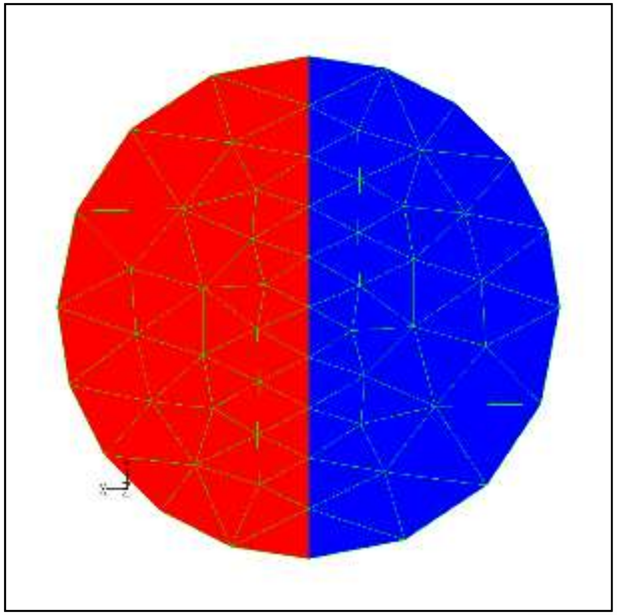
Numerical diffusion



Block structured

Cell count = 7400

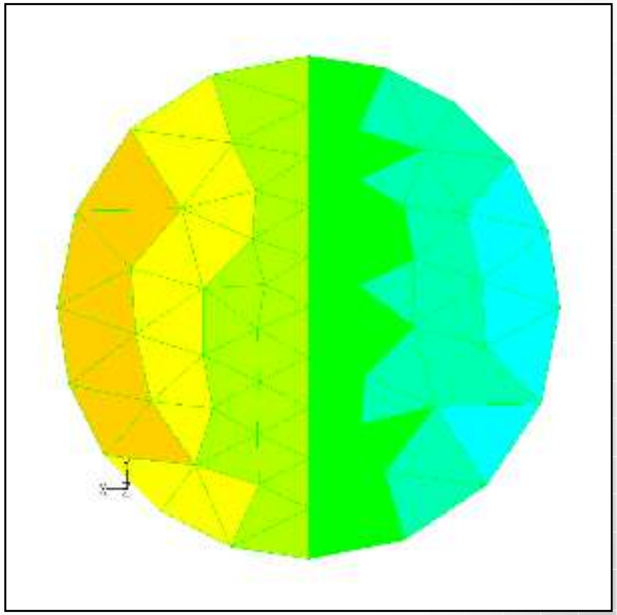
Aligned with flow



Cooper prismatic grid

Cell count = 9800

Aligned with flow



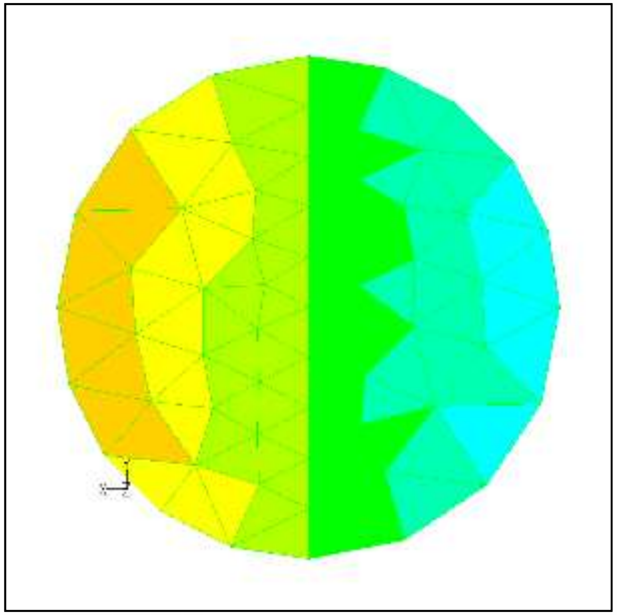
Unstructured grid

Cell count = 29146

Not aligned with flow

Introduction

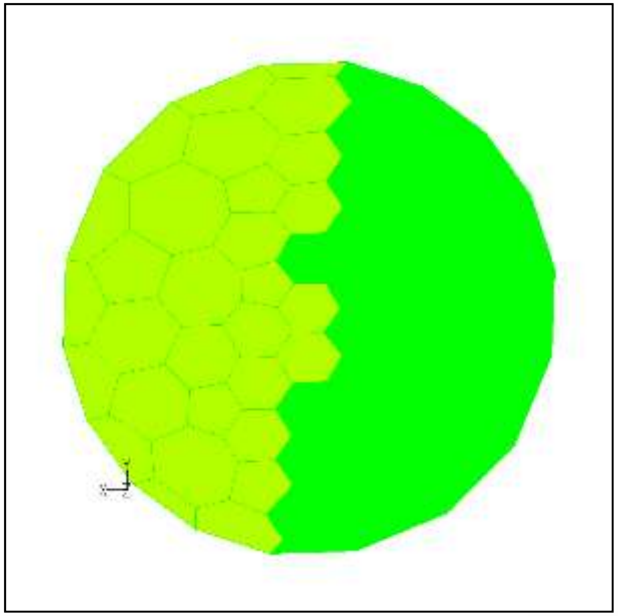
Numerical diffusion



Unstructured grid

Cell count = 29146

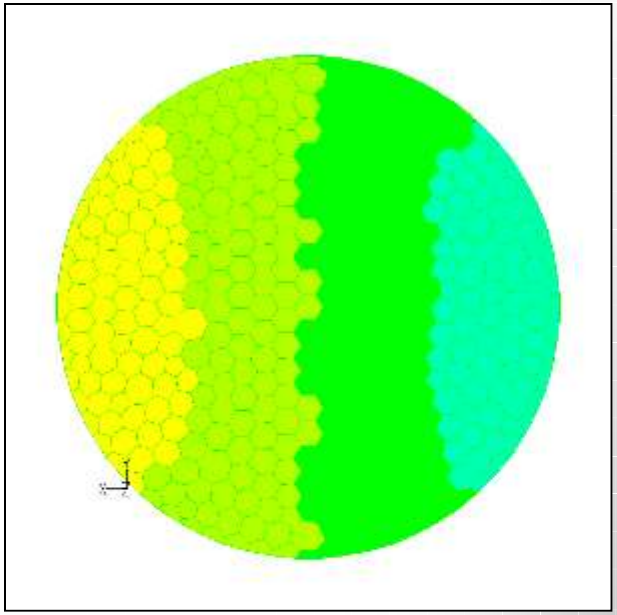
Not aligned with flow



Polyhedral grid

Cell count = 8116

Not aligned with flow



(Refined, 0.05m) Polyhedral grid

Cell count = 72589

Not aligned with flow

Introduction

- However…
- CFD approach is restricted by computing resource available (although this is becoming less of a constraint)
- CFD is a first-principles approach which requires **validation**.

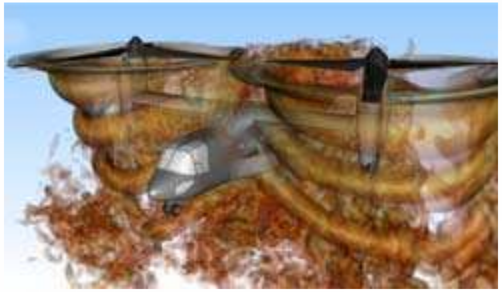
Agenda

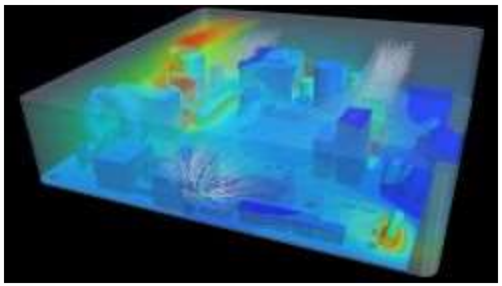
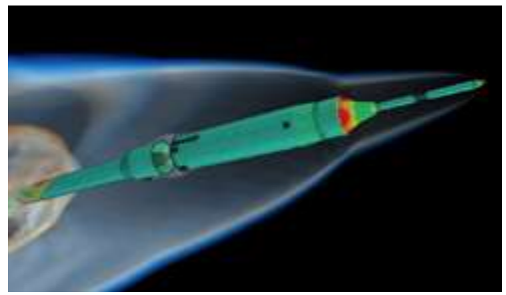
- Introduction
- Industrial application of CFD
- Lower cost and open source simulation tools
- Verification and validation
- Summary.

Agenda

- Industrial application of CFD
  - General examples (Star-CCM+)
  - Technical safety
  - Subsea hydrodynamics
  - Thermal analysis
  - Flow assurance
  - Vortex/flow induced vibration
  - Tidal flow.

Industrial application of CFD

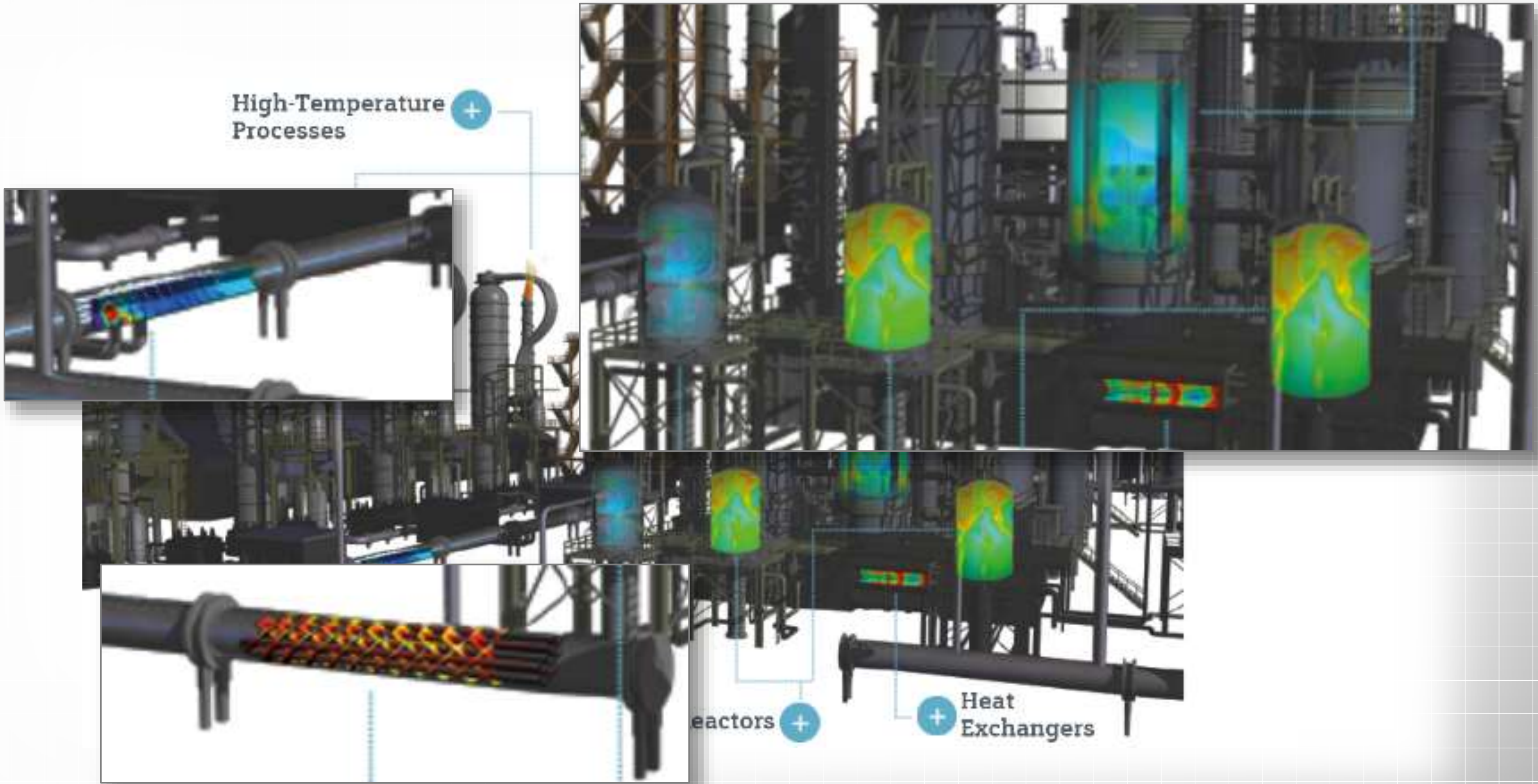




Siemens Star-CCM+: http://mdx.plm.automation.siemens.com/star-ccm-plus

Industrial application of CFD



Siemens Star-CCM+: http://mdx.plm.automation.siemens.com/star-ccm-plus

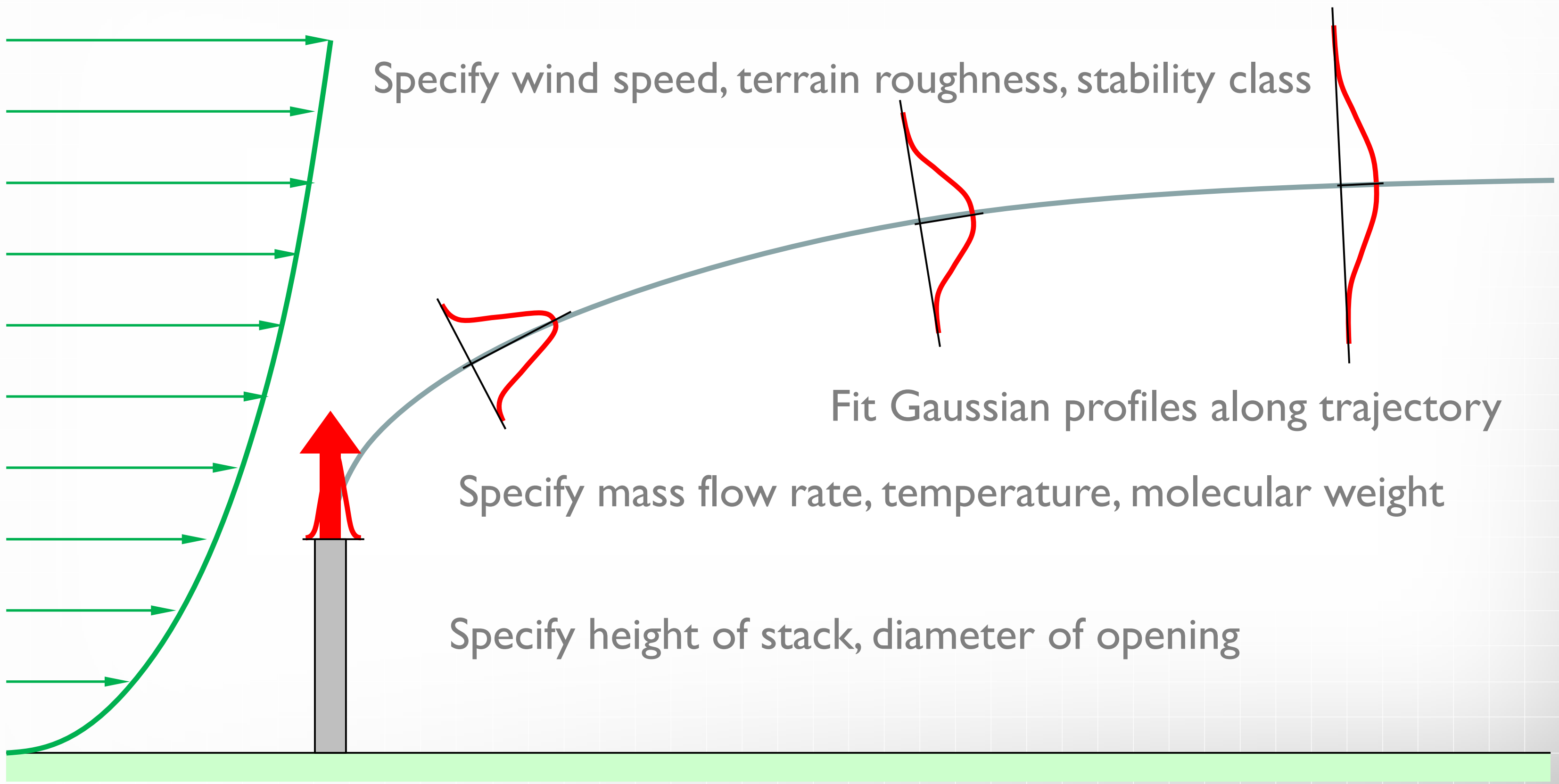
Industrial application of CFD

Atmospheric dispersion

Offshore helideck design guidelines, HSE.

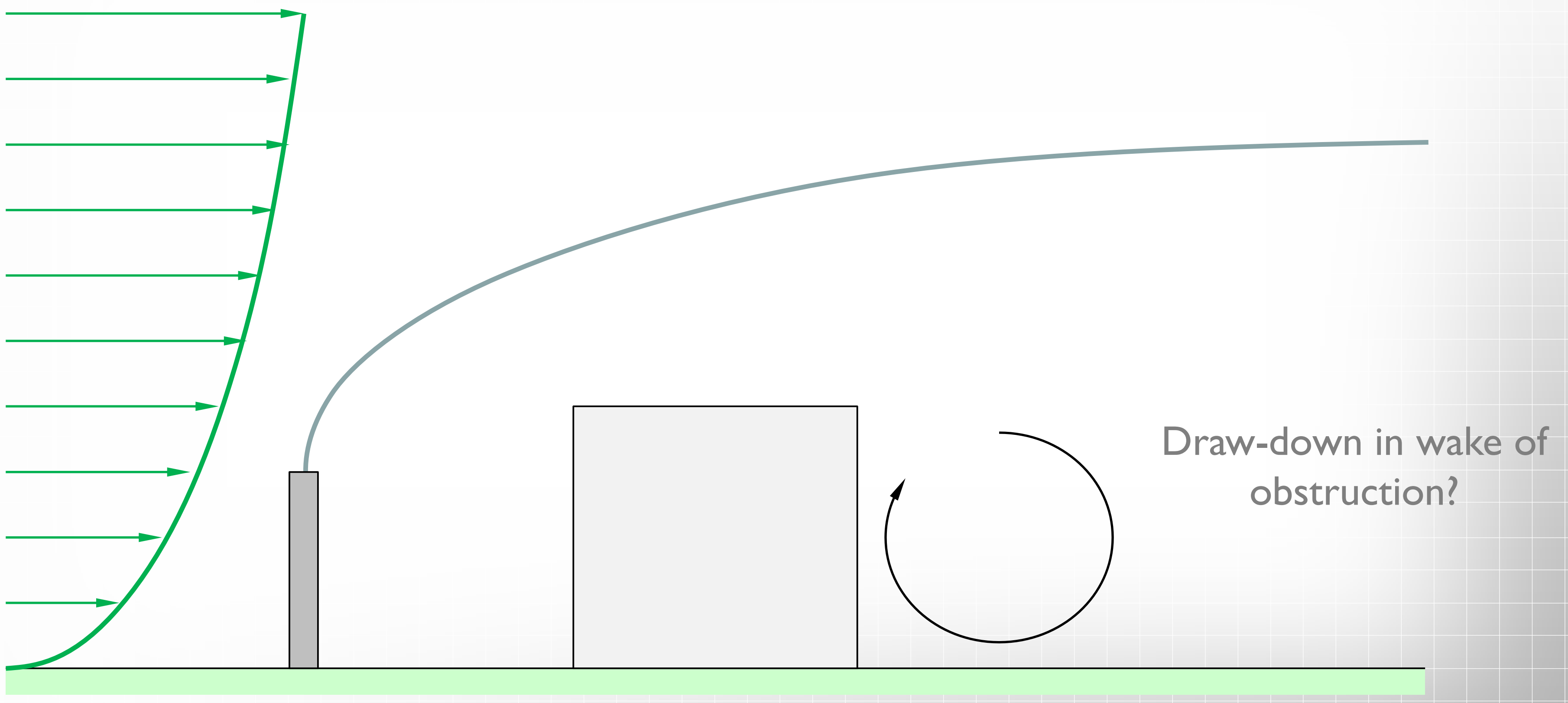
Industrial application of CFD

Atmospheric dispersion



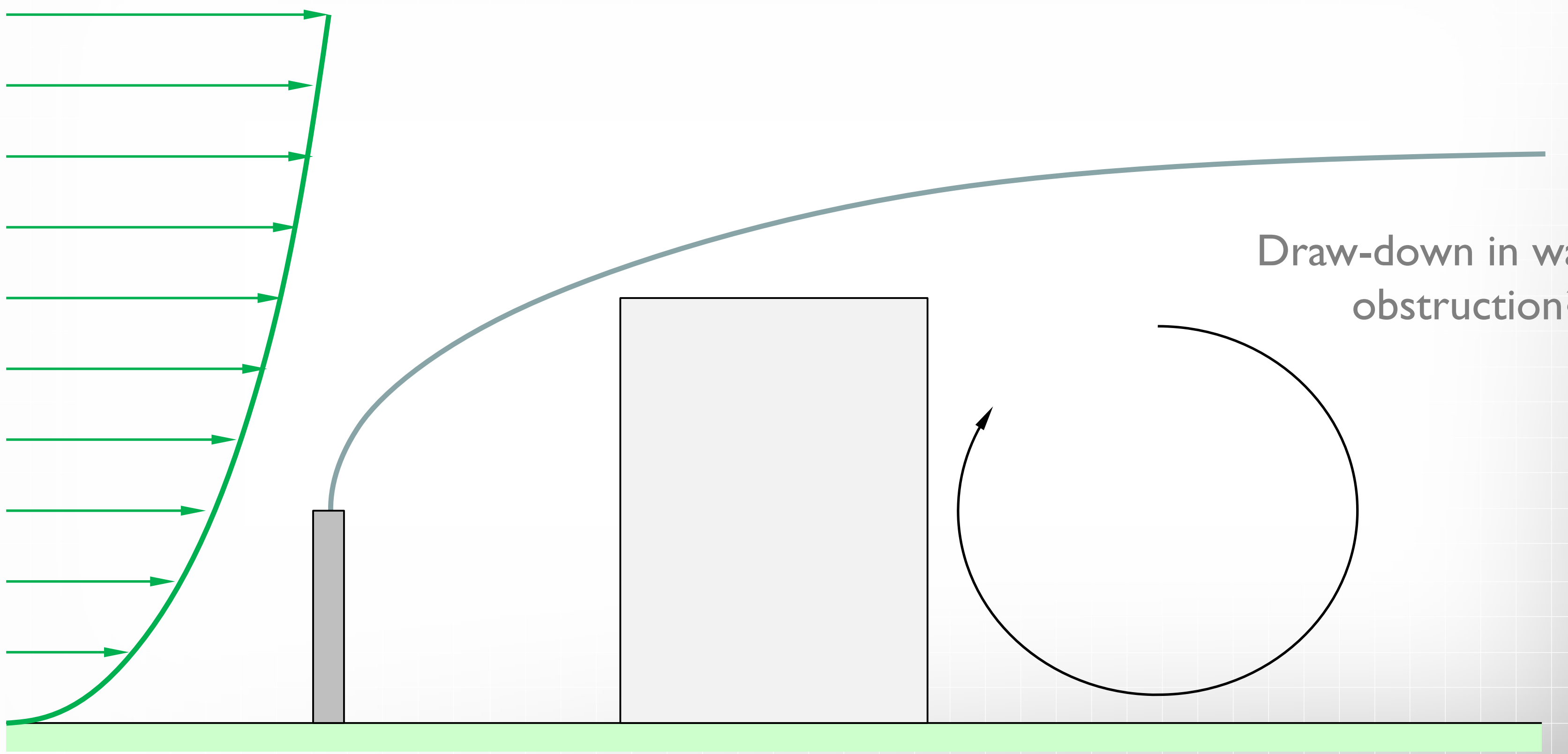
Industrial application of CFD

Atmospheric dispersion



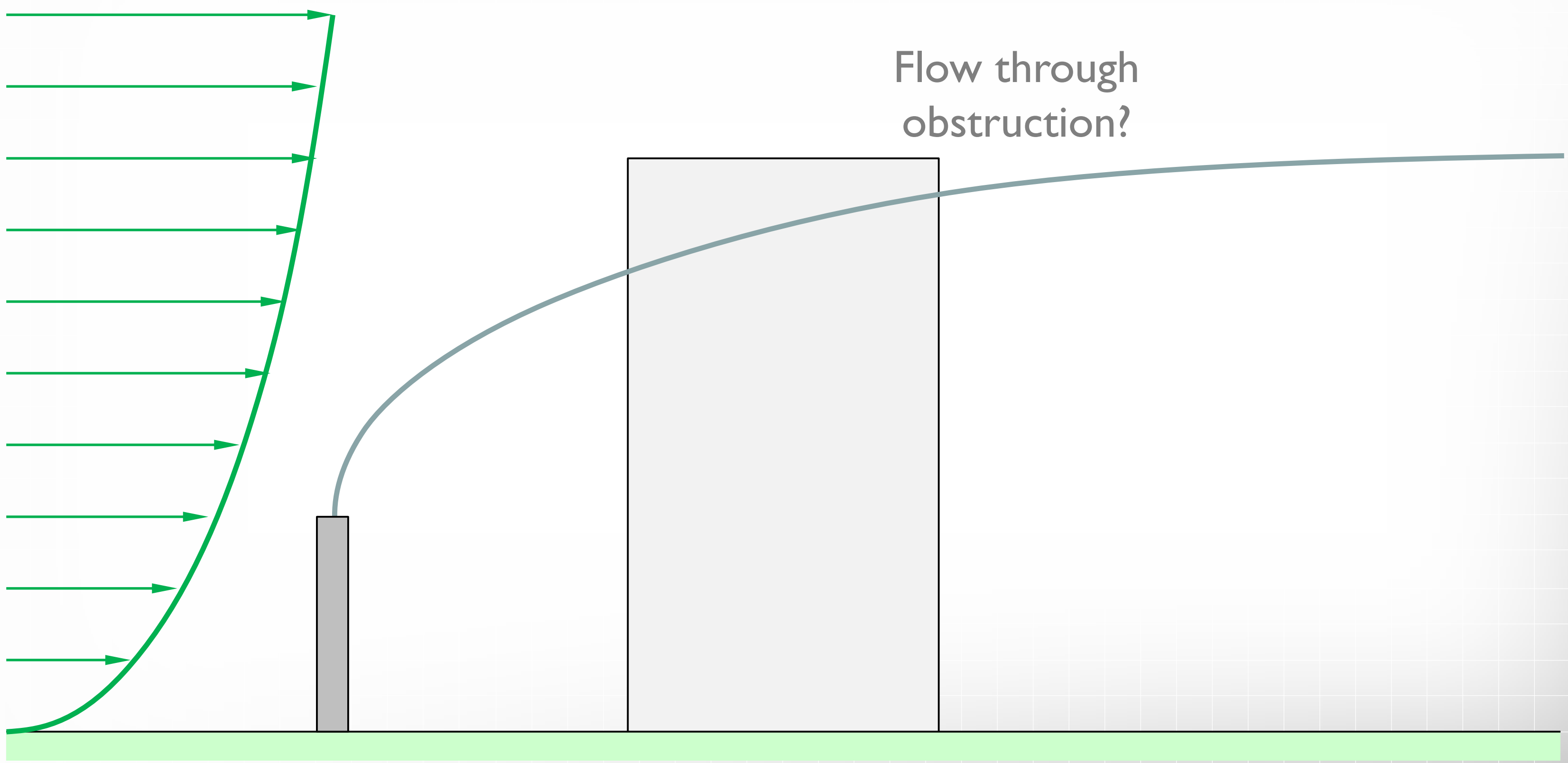
Industrial application of CFD

Atmospheric dispersion



Industrial application of CFD

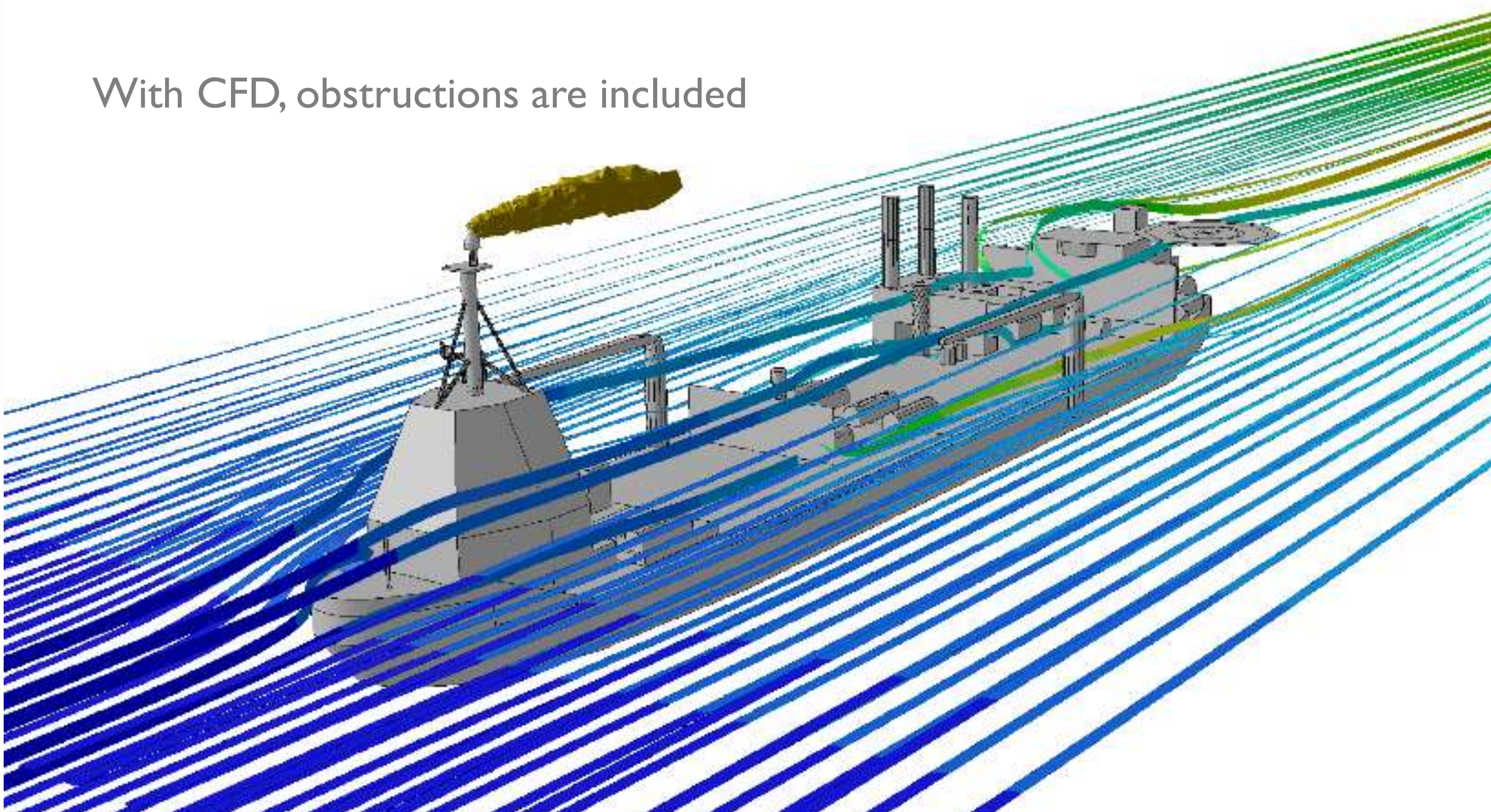
Atmospheric dispersion



Industrial application of CFD

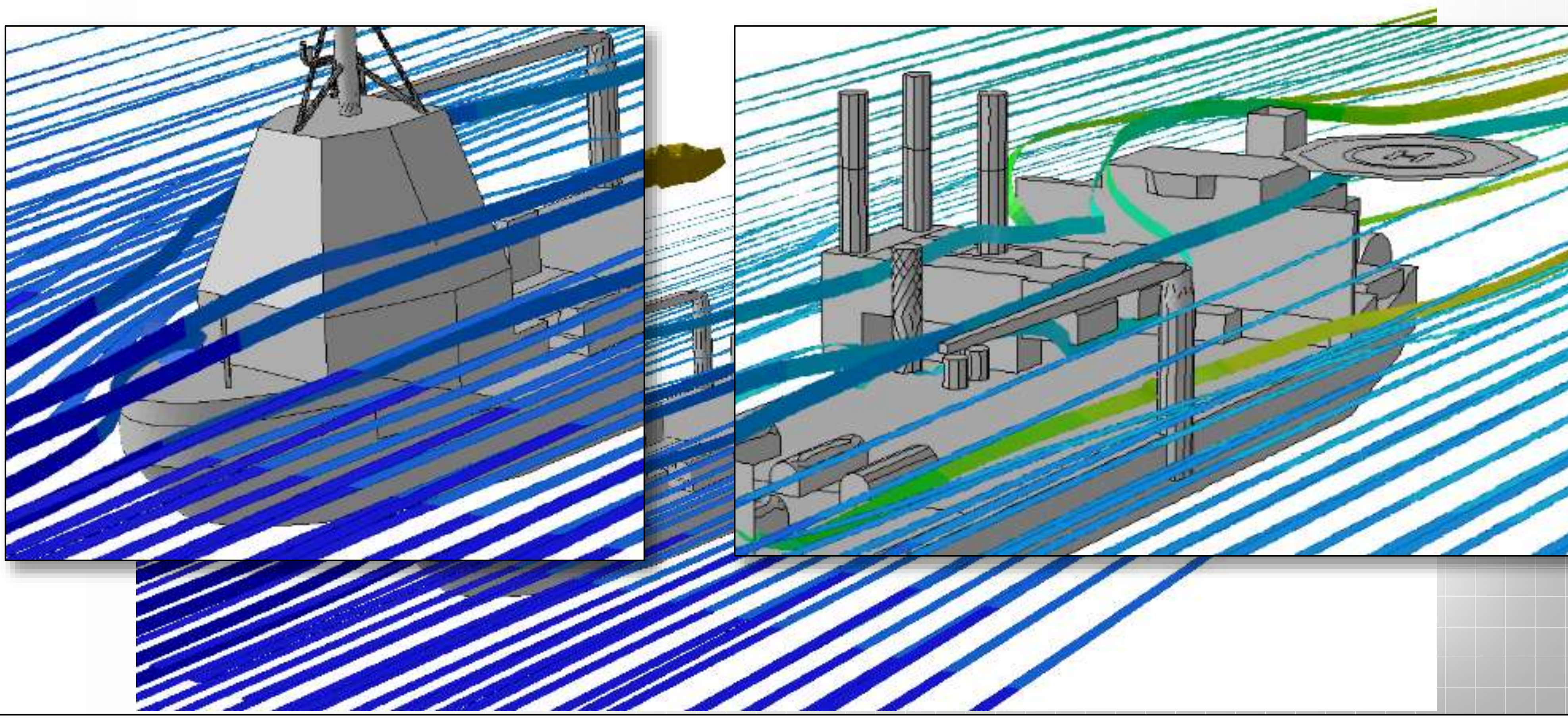
Atmospheric dispersion

With CFD, obstructions are included



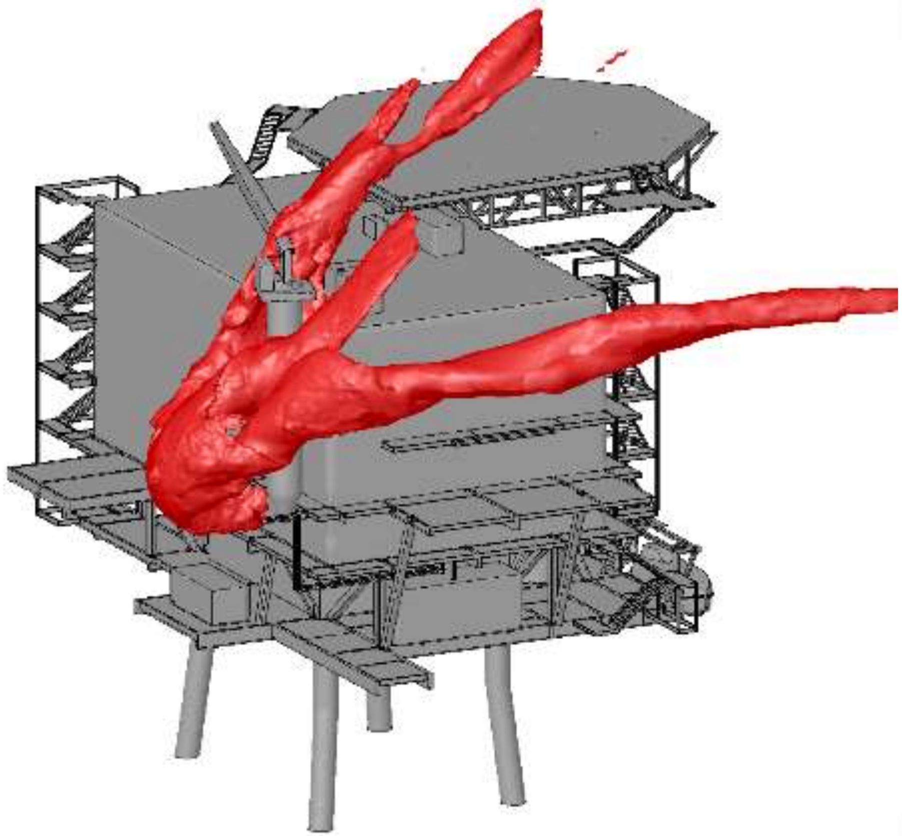
Industrial application of CFD

Atmospheric dispersion



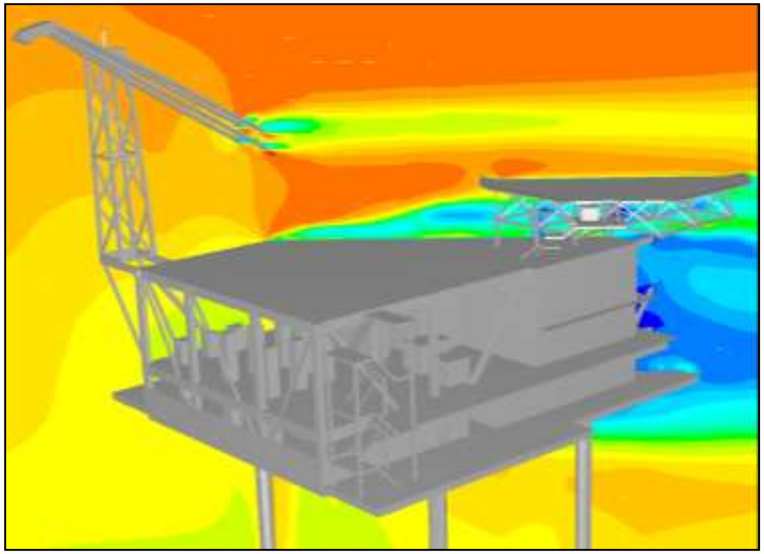
Industrial application of CFD

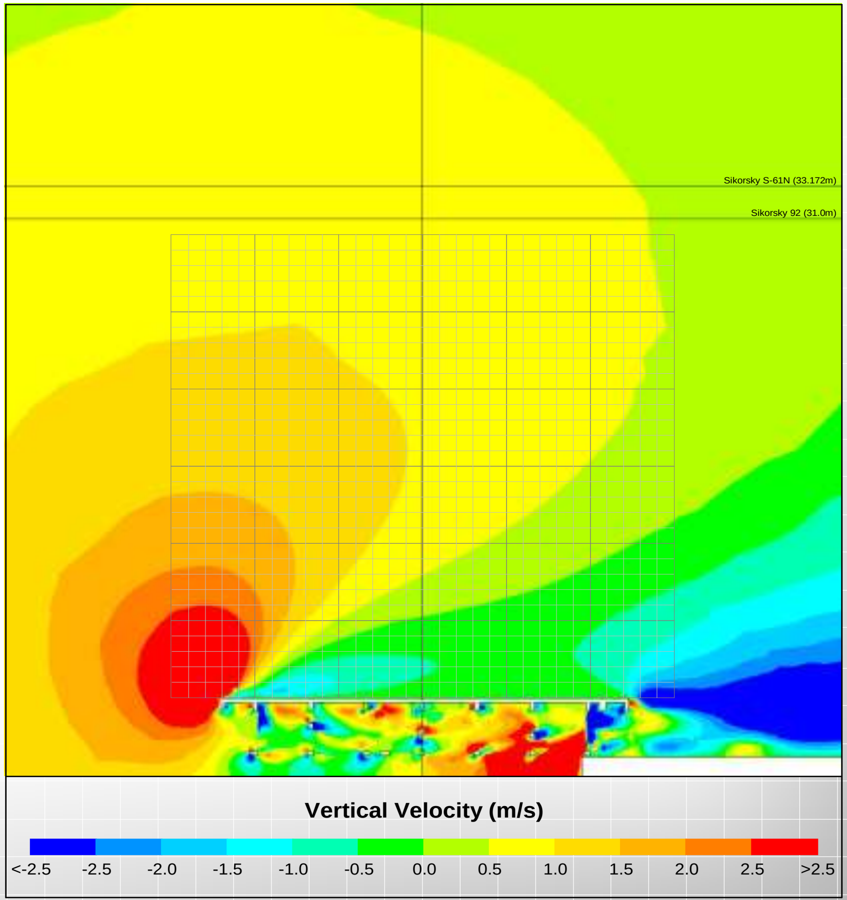
Atmospheric dispersion



Industrial application of CFD

Helideck environment

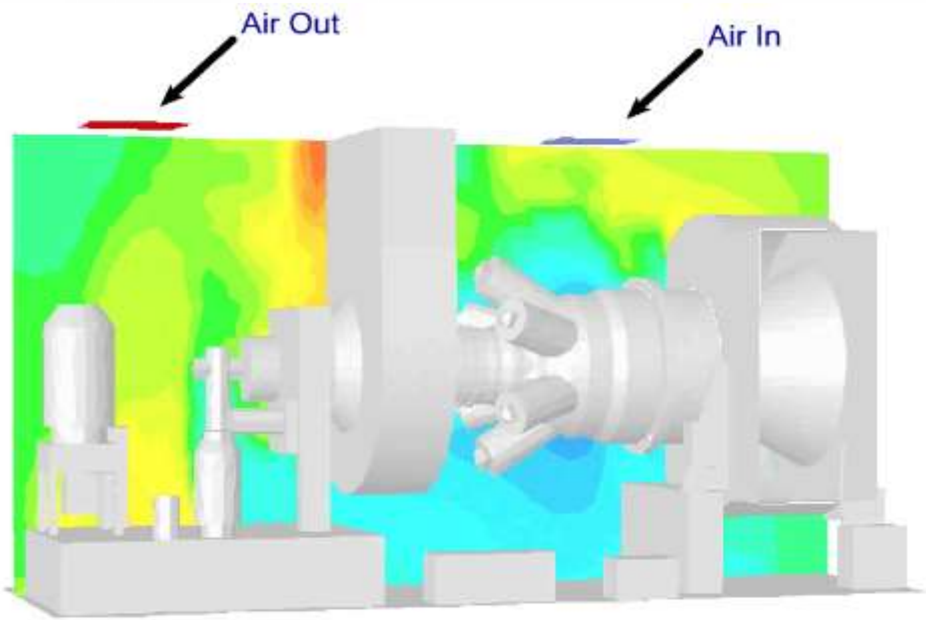


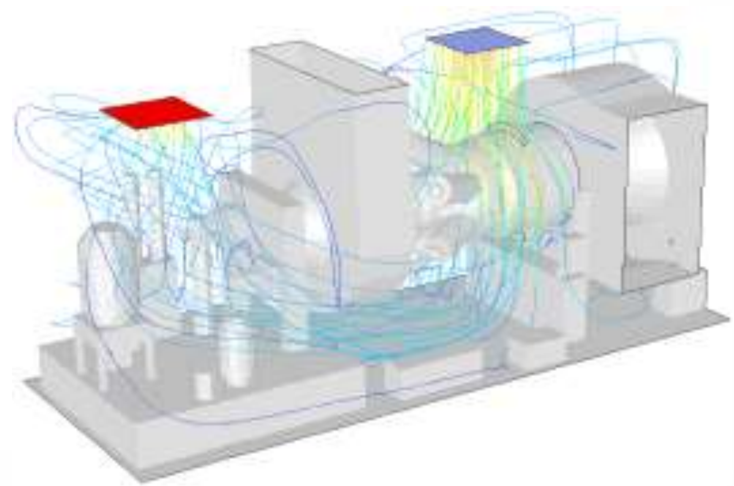


Prediction of velocity field and turbulence fluctuations over the helideck

Industrial application of CFD

Ventilation performance

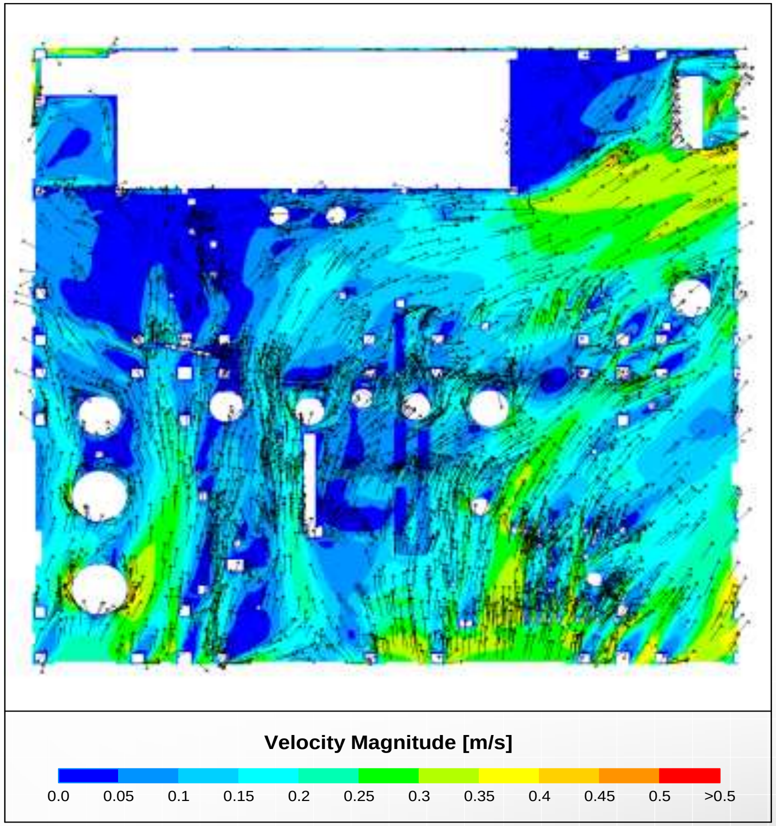


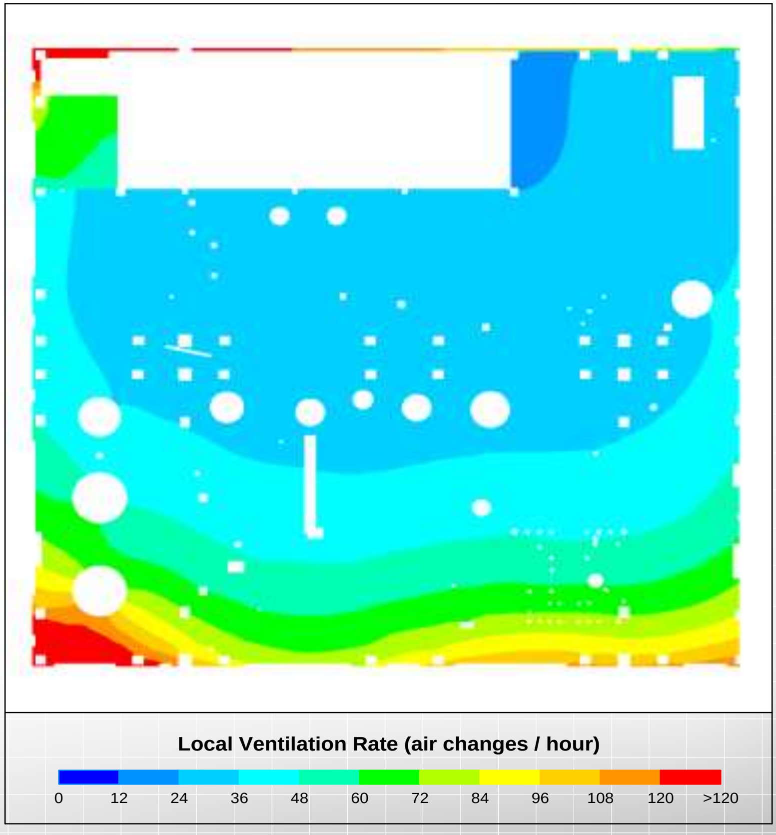


Sections Through A Turbine Enclosure Showing Mean Age of Air
(Red = Old Air, Blue = New Air)

Industrial application of CFD

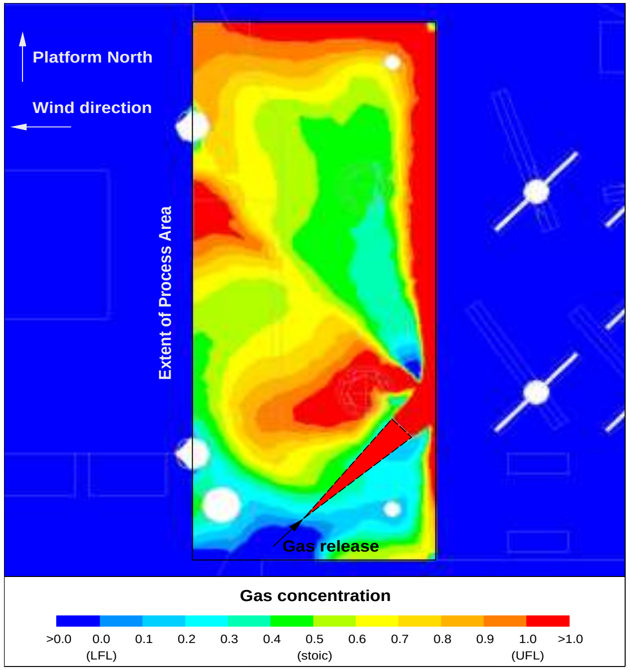
Ventilation performance



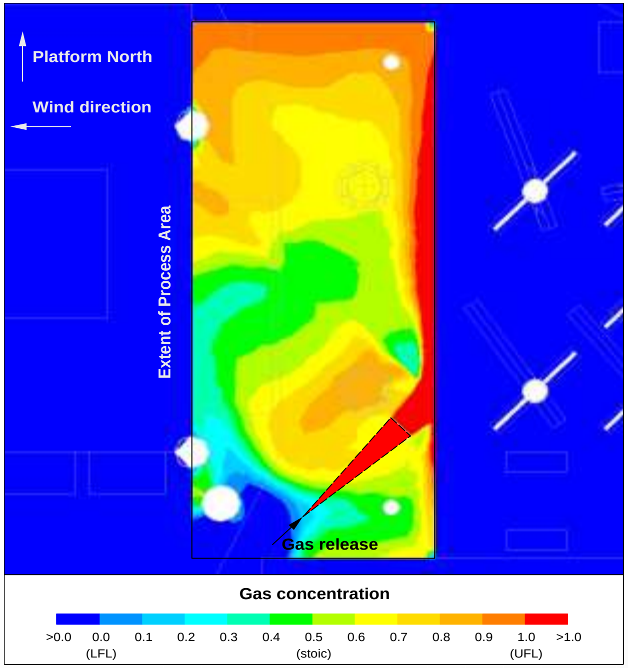


Industrial application of CFD

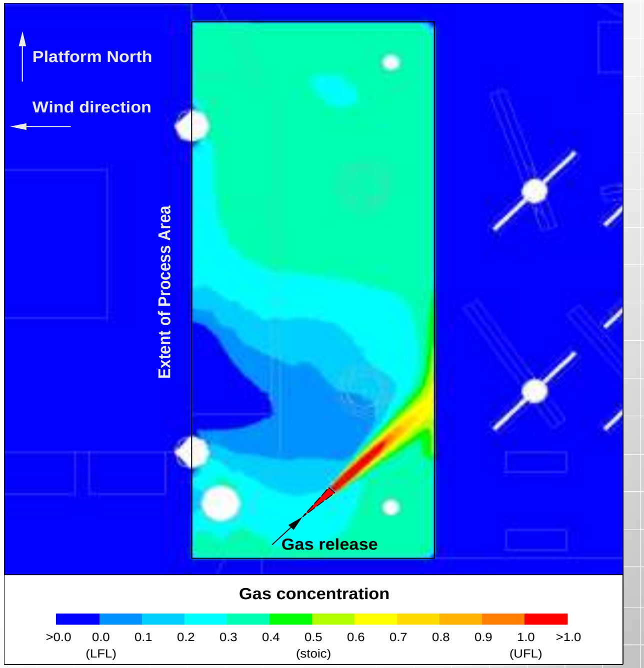
Gas leak dispersion



Large release



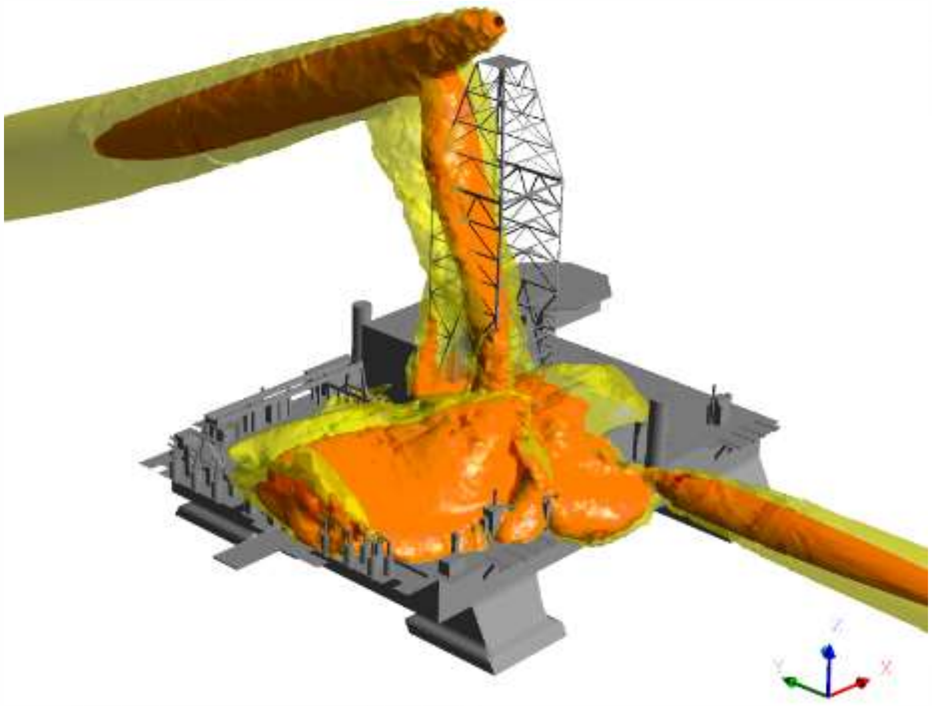
Medium release

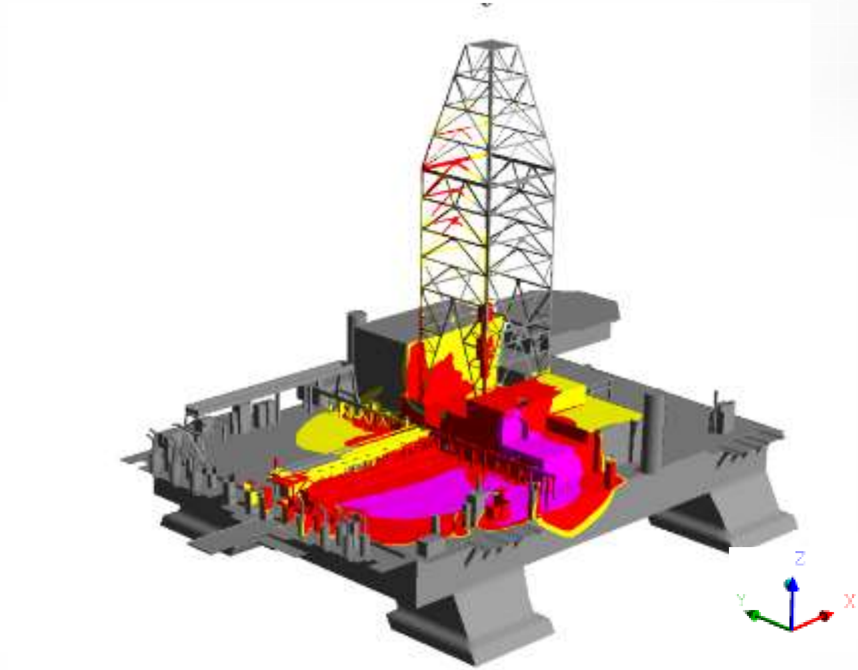


Small release

Industrial application of CFD

Gas leak dispersion





Deepwater Horizon investigation: http://abercus.com/News_20110622.aspx

Industrial application of CFD

Explosions

(Courtesy of Gexcon)

- Both configurations contain the same volume of gas and volumetric fill of pipe work
- The configuration on the left comprises a few large diameter pipes
- The configuration on the right comprises many small diameter pipes
- The intensity of the explosion for the right-hand configuration is increased significantly.

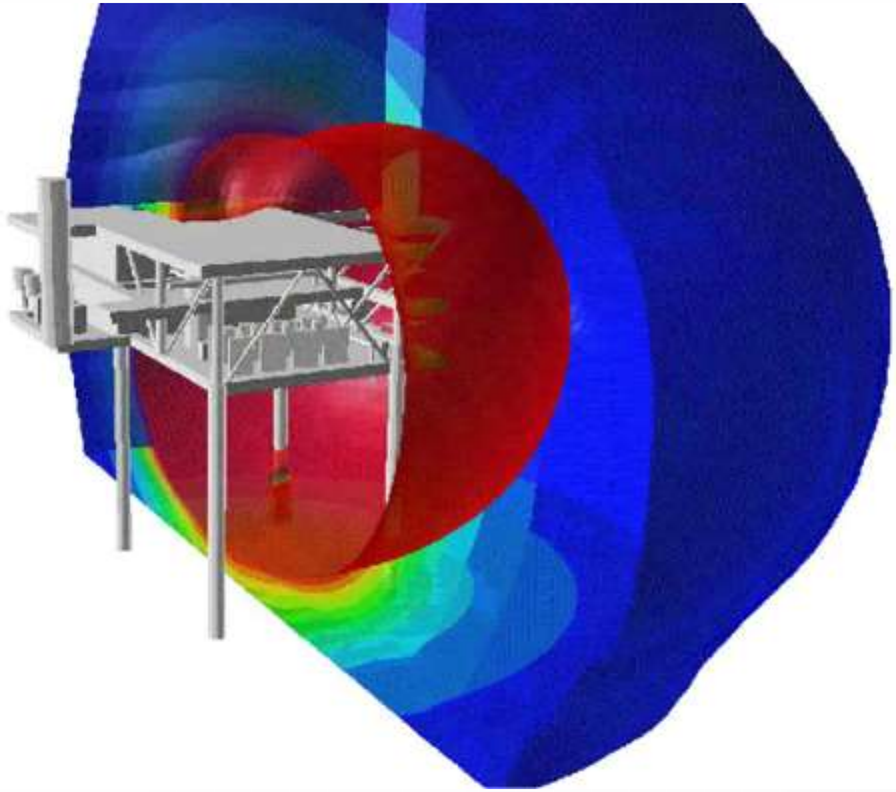
Industrial application of CFD

Explosions

- Both configurations contain the same volume of gas
- The configuration on the left is entirely filled with small-scale congestion
- The configuration on the right is half-filled with small-scale congestion.

Industrial application of CFD

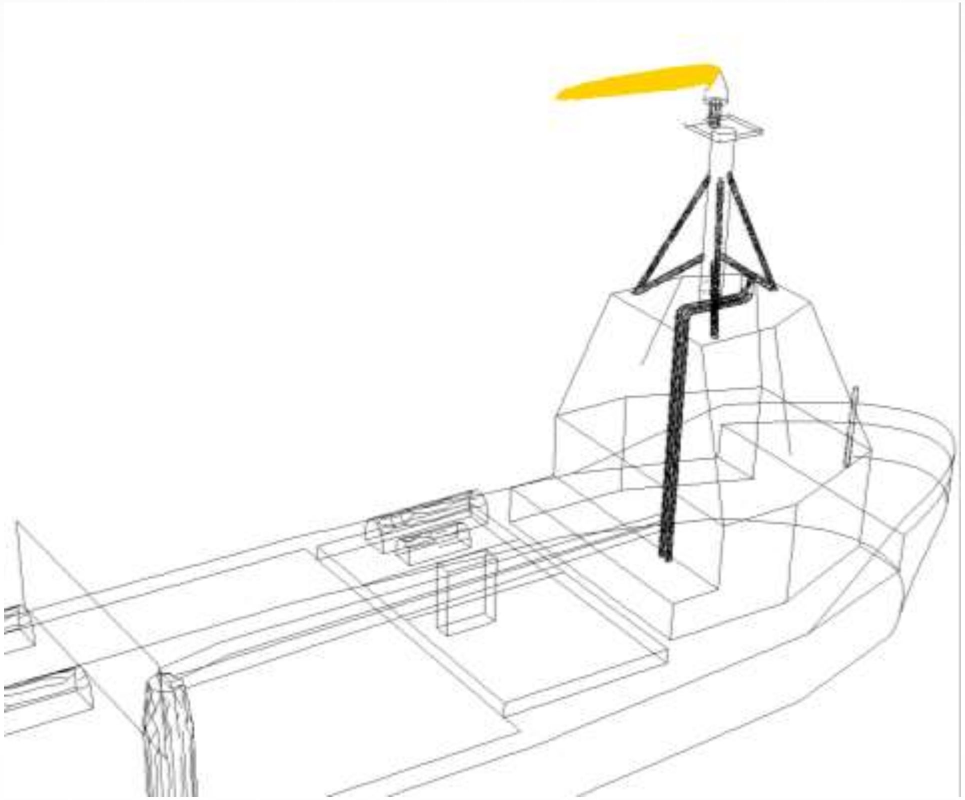
Explosions



Abercus Riverside House Riverside Drive Aberdeen AB11 7LH www.abercus.com

Industrial application of CFD

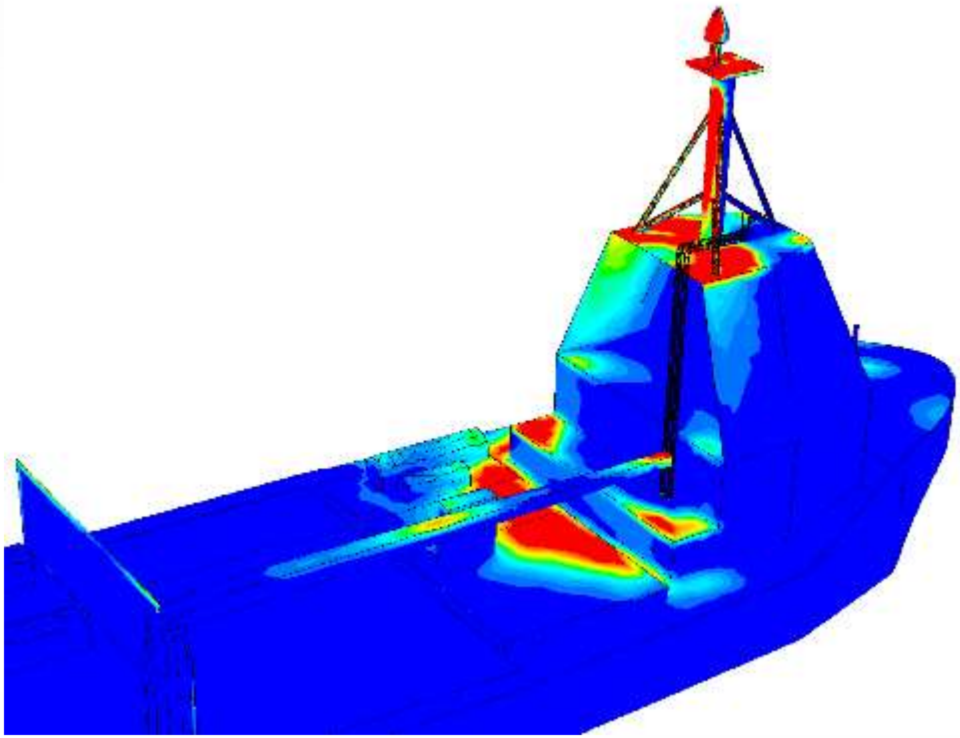
Flaring



Envelope of combustion zone

Industrial application of CFD

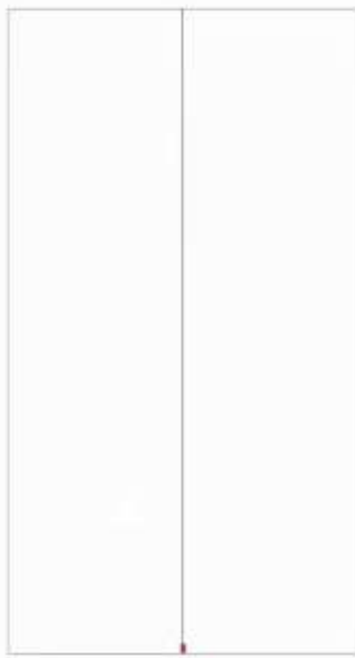
Flaring



Incident radiation

Industrial application of CFD

Subsea releases



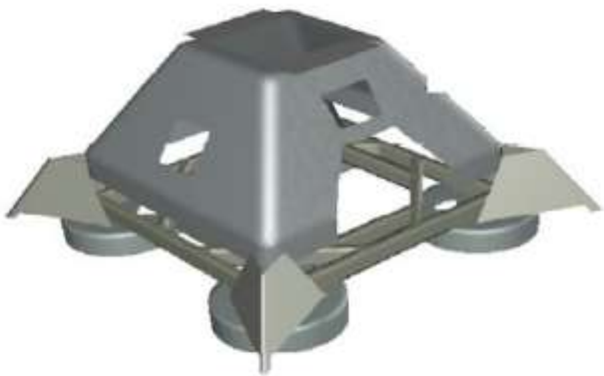
Sustained release

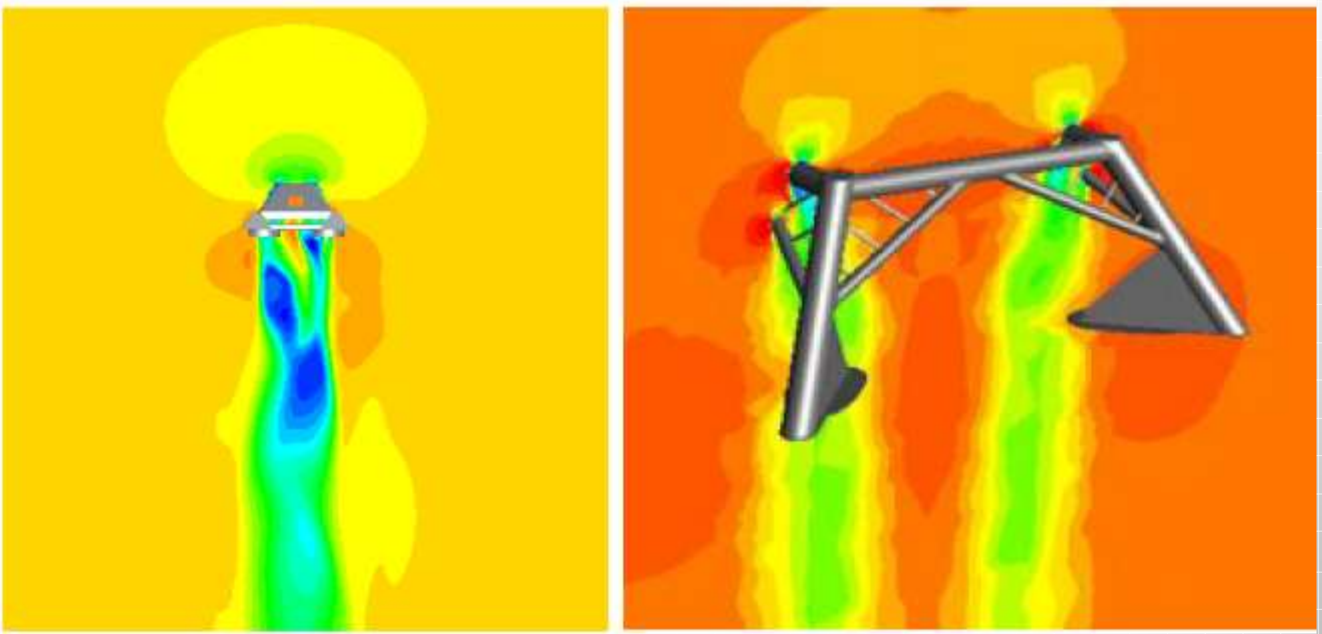
Pulsed release

Gas concentration [% volume at atmospheric pressure]

Industrial application of CFD

Drag and added mass

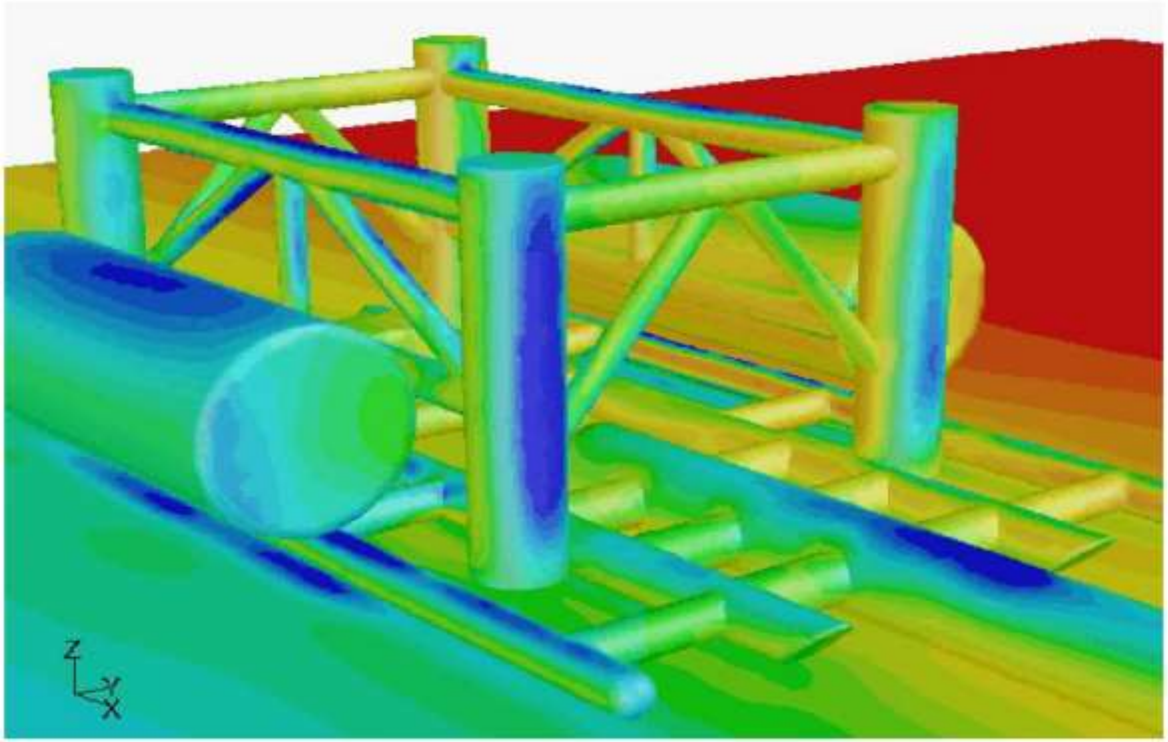




Velocity magnitude

Industrial application of CFD

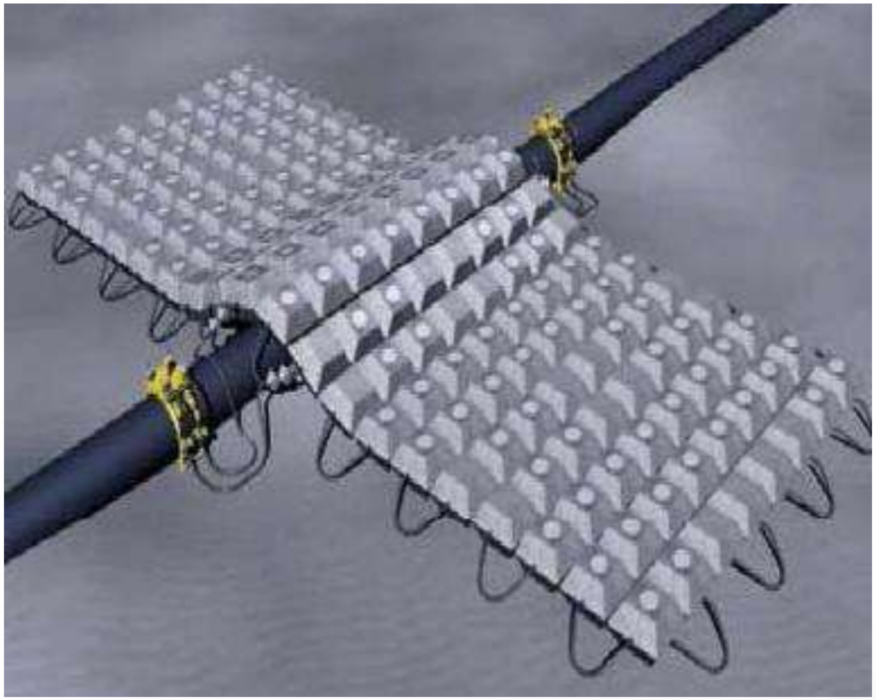
Stability

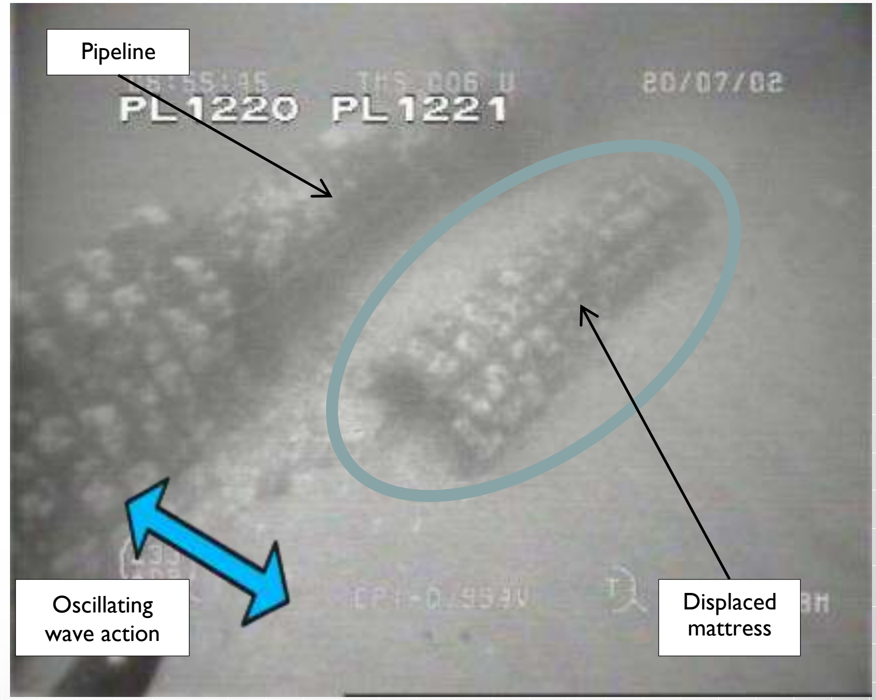


Surface pressure

Industrial application of CFD

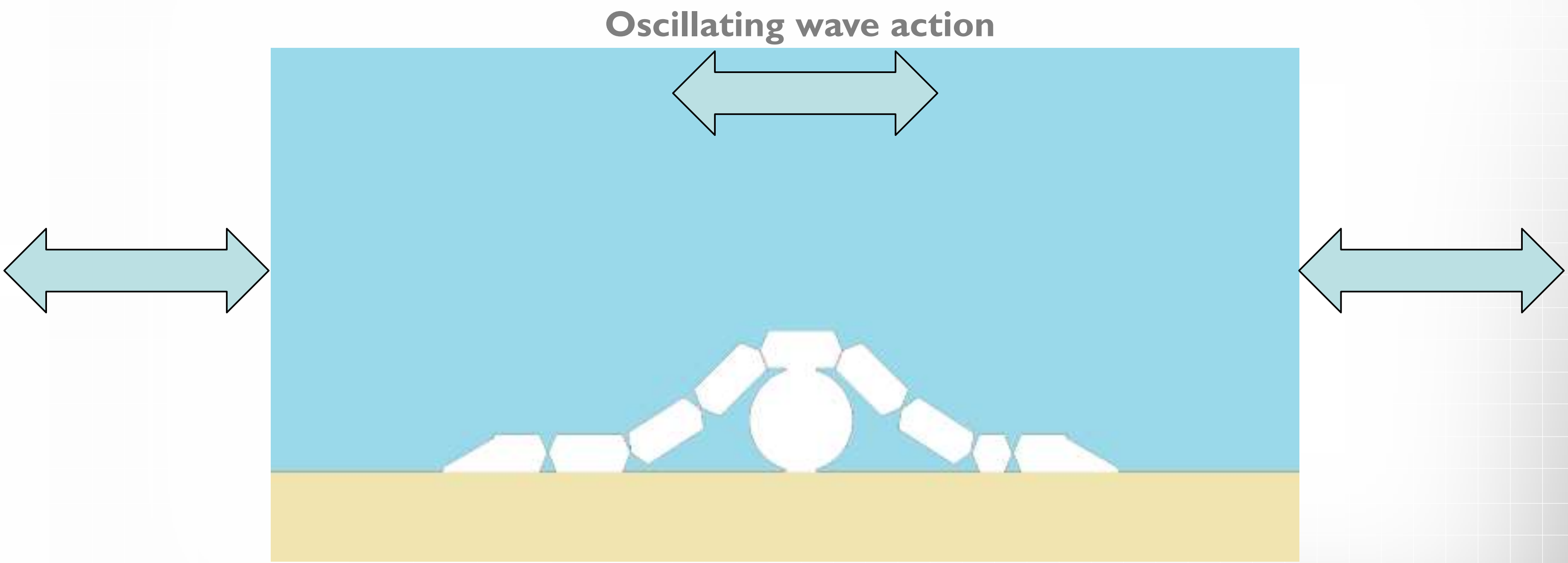
Stability





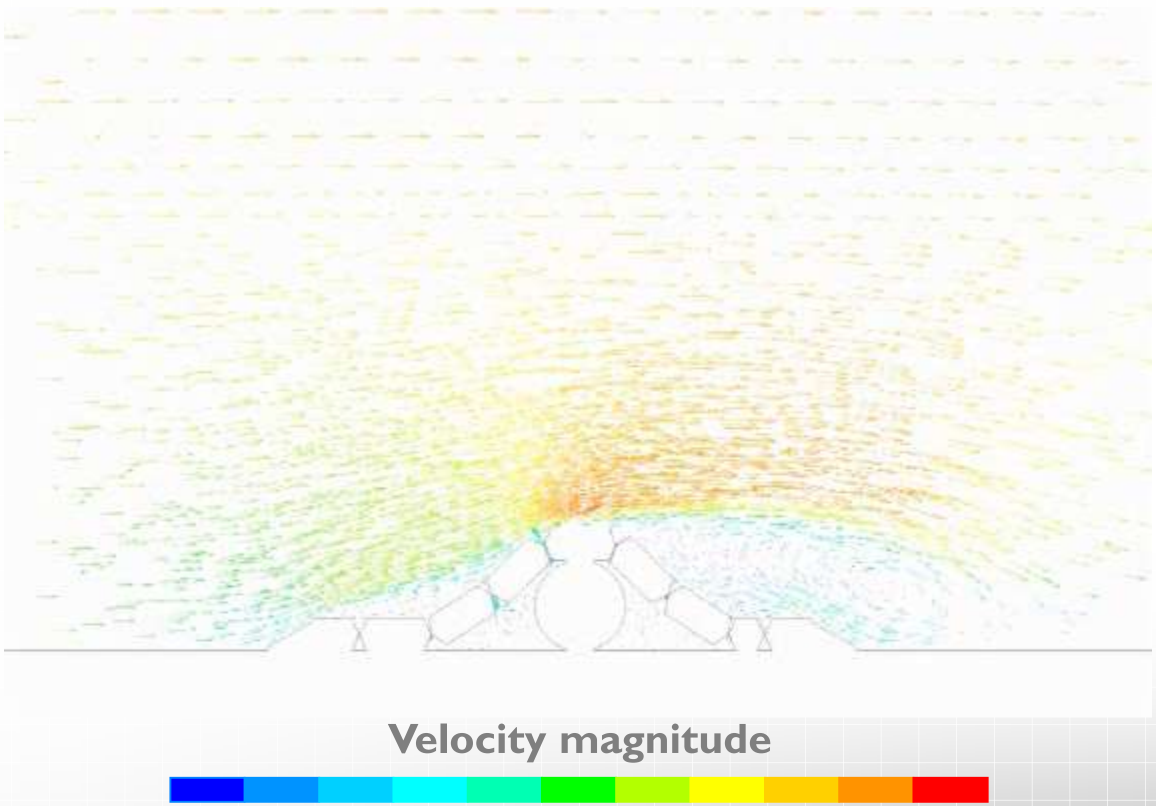
Industrial application of CFD

Stability



Industrial application of CFD

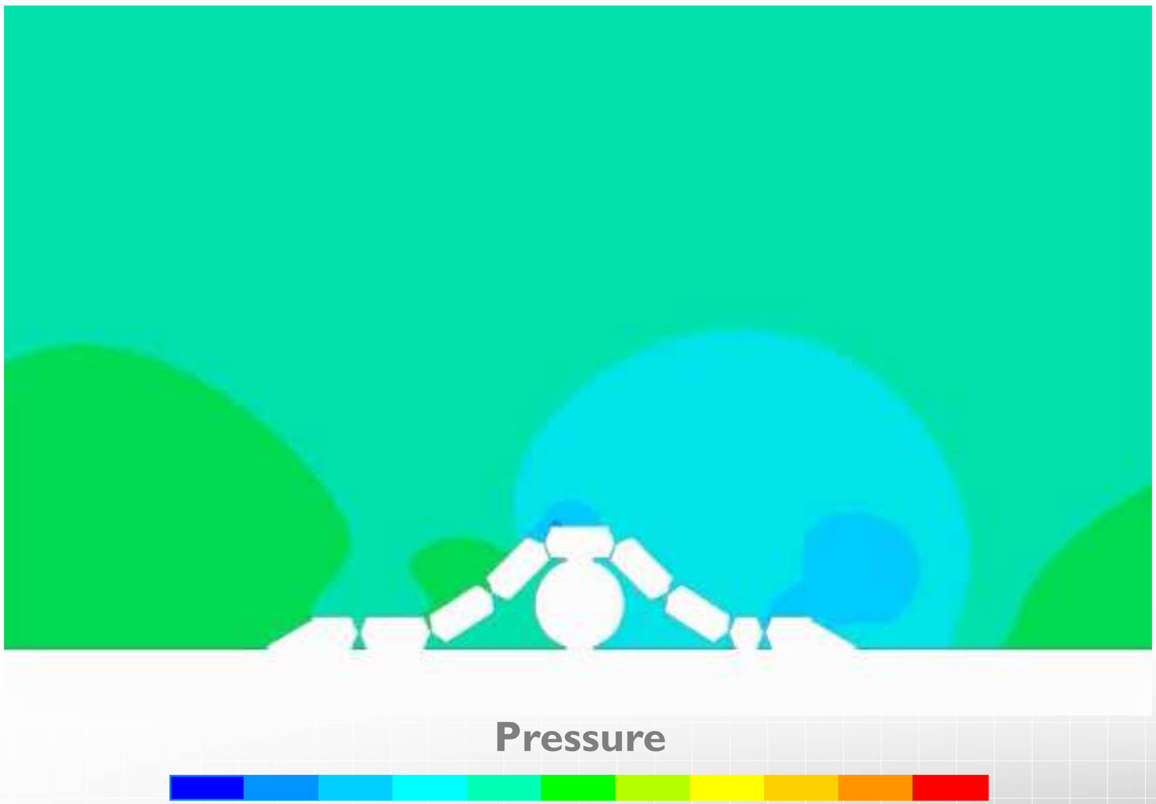
Stability



Velocity magnitude

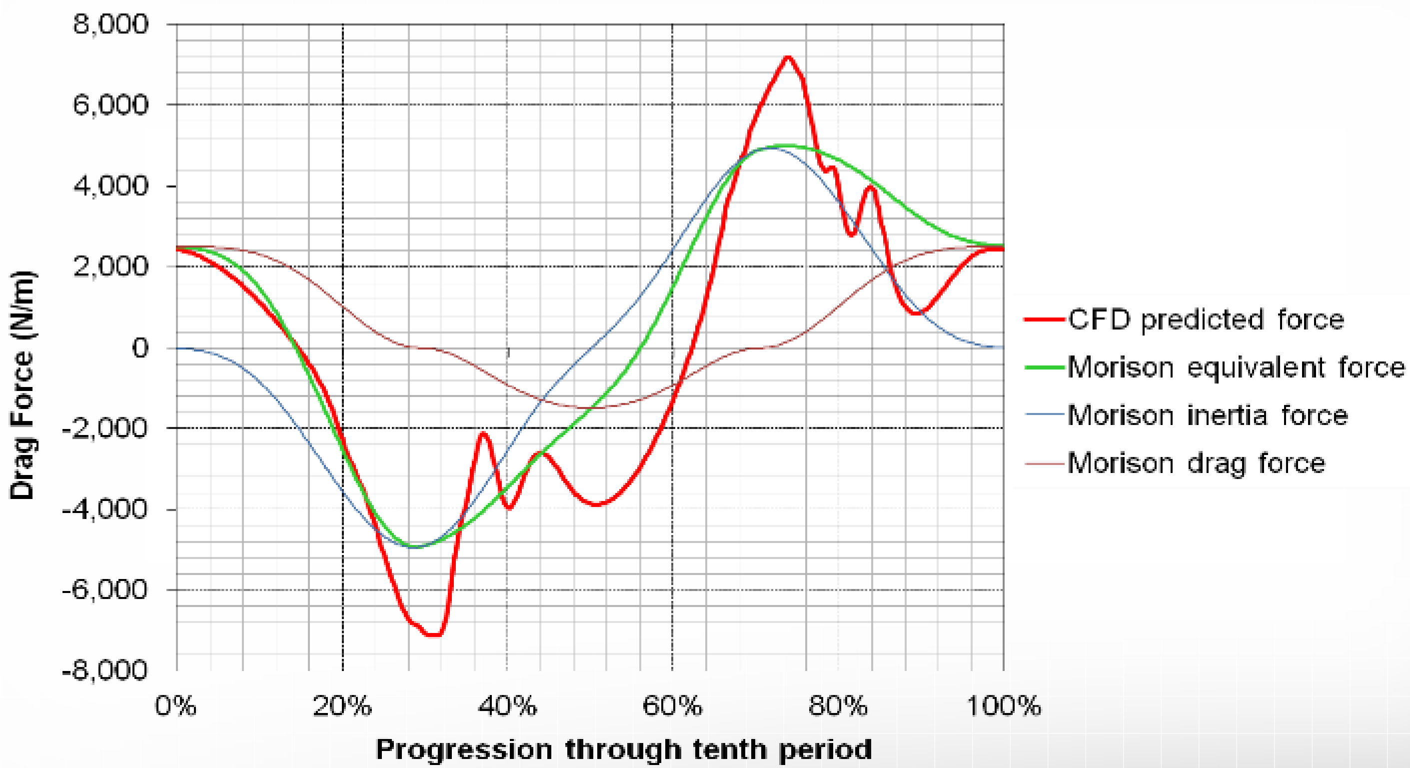
Industrial application of CFD

Stability



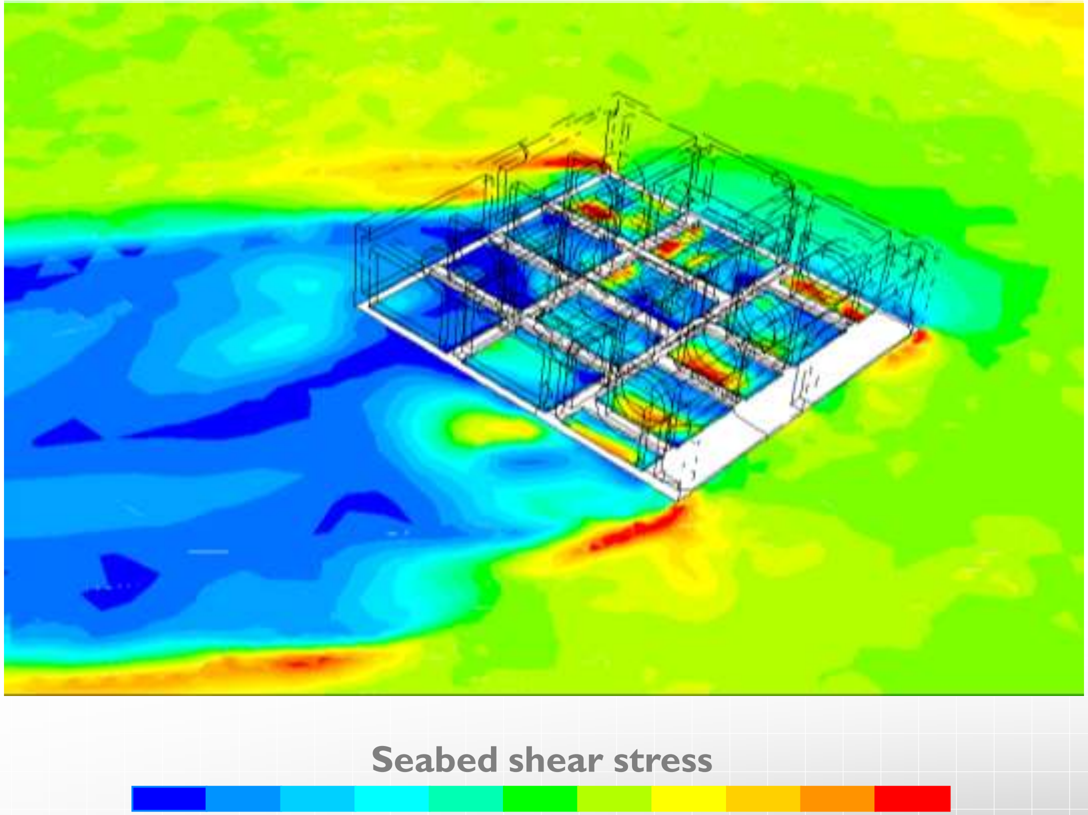
Industrial application of CFD

Stability



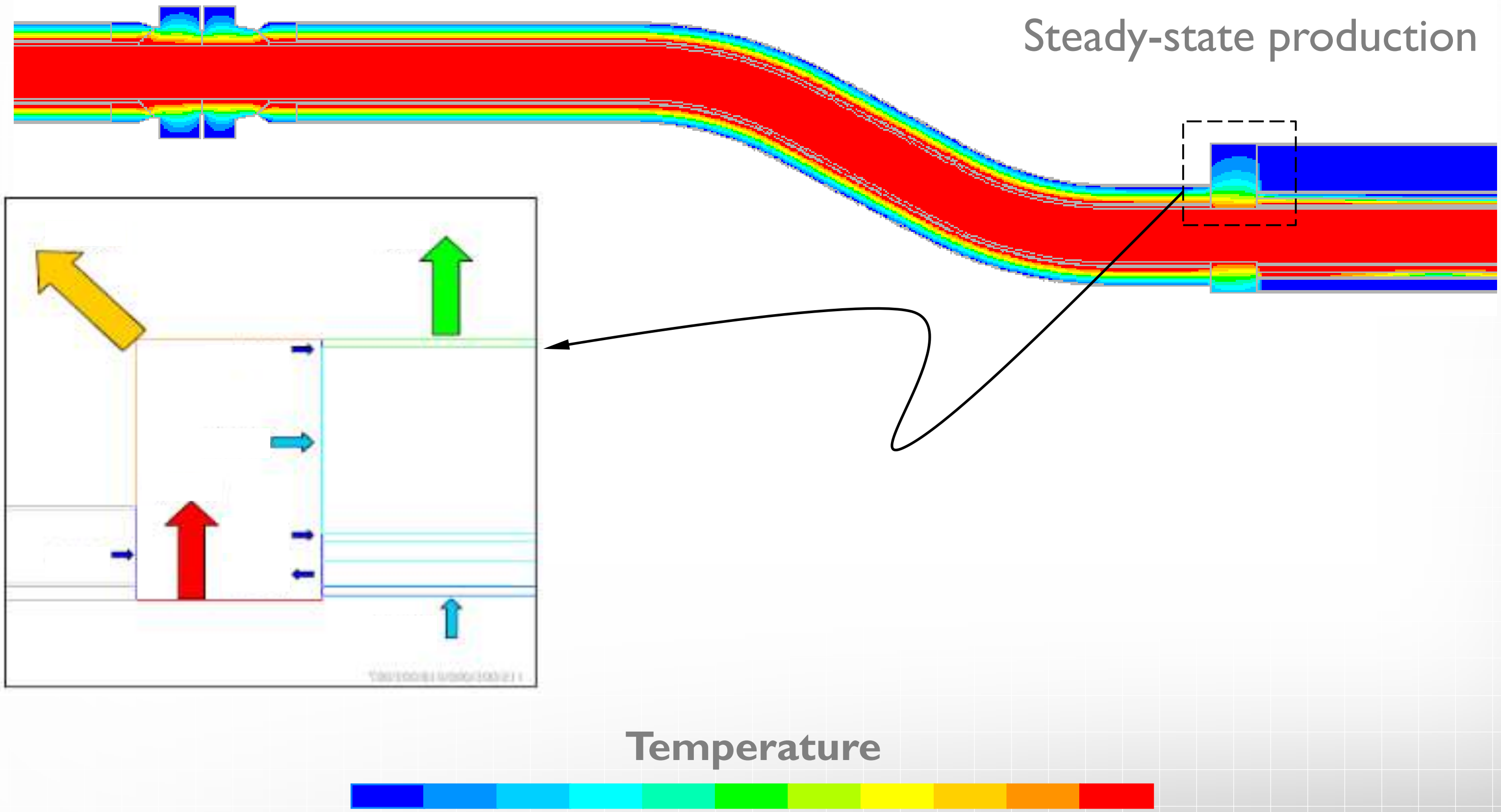
Industrial application of CFD

Scour



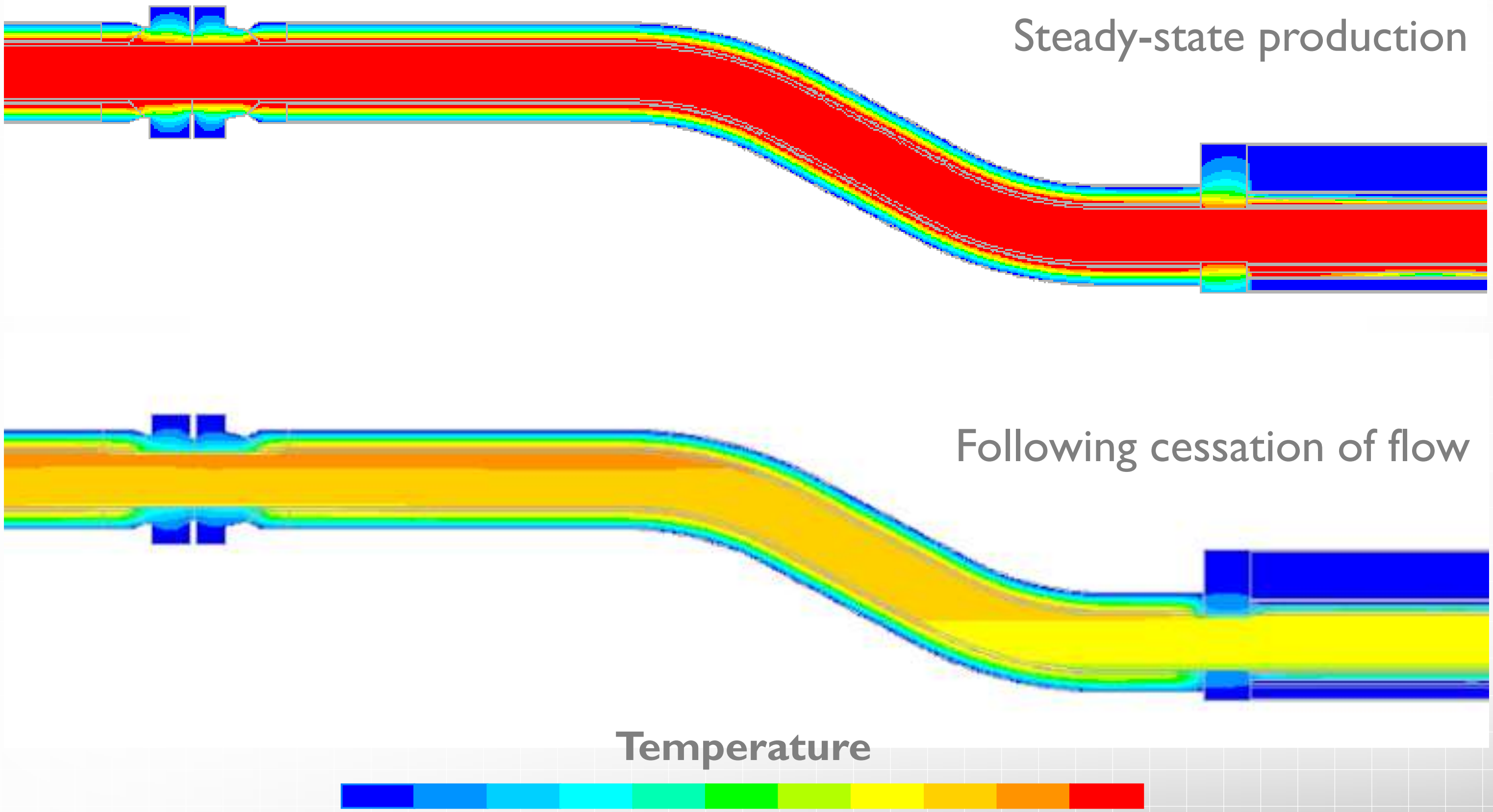
Industrial application of CFD

Thermal analysis



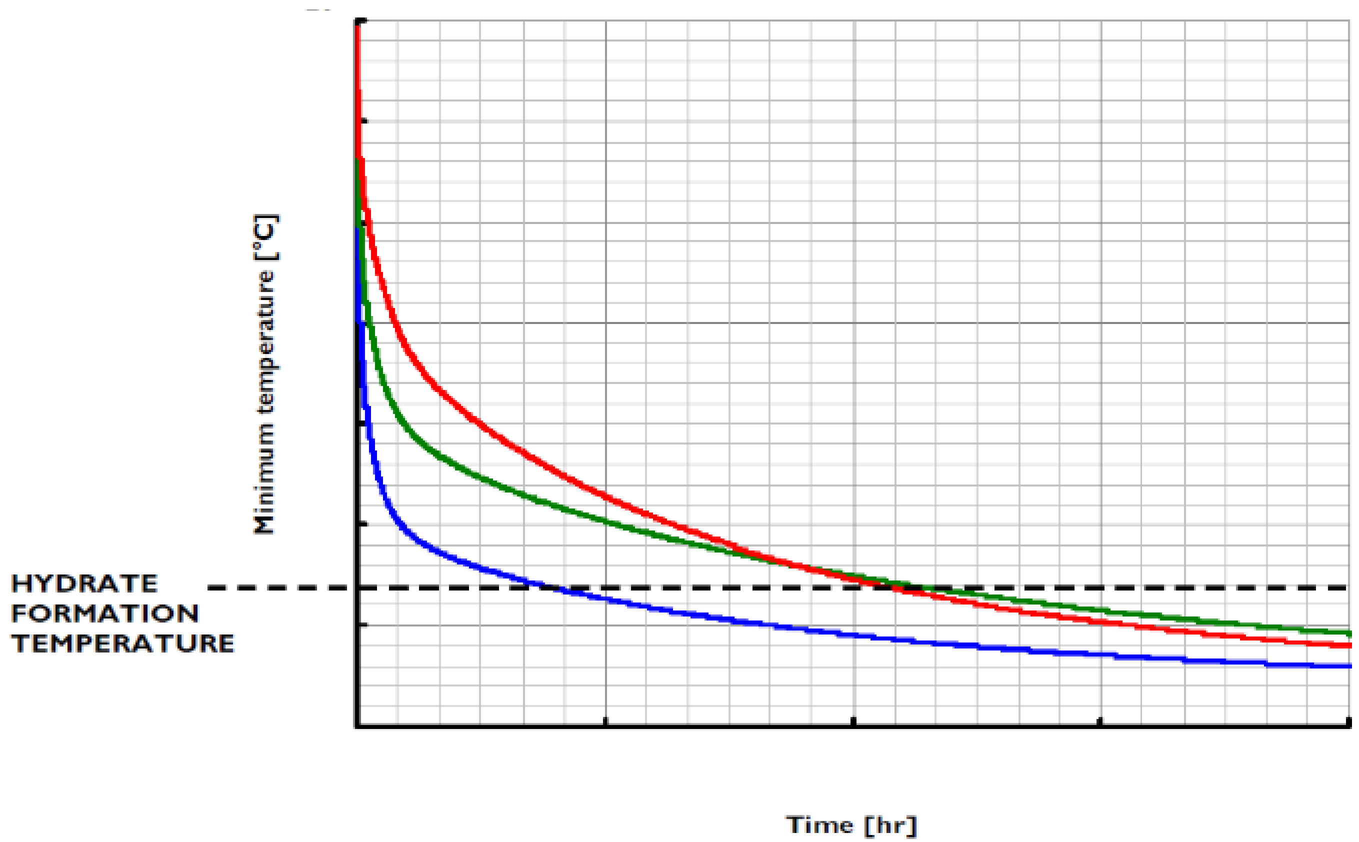
Industrial application of CFD

Thermal analysis



Industrial application of CFD

Thermal analysis



Industrial application of CFD

Thermal analysis

Gasket materi
Casing materia

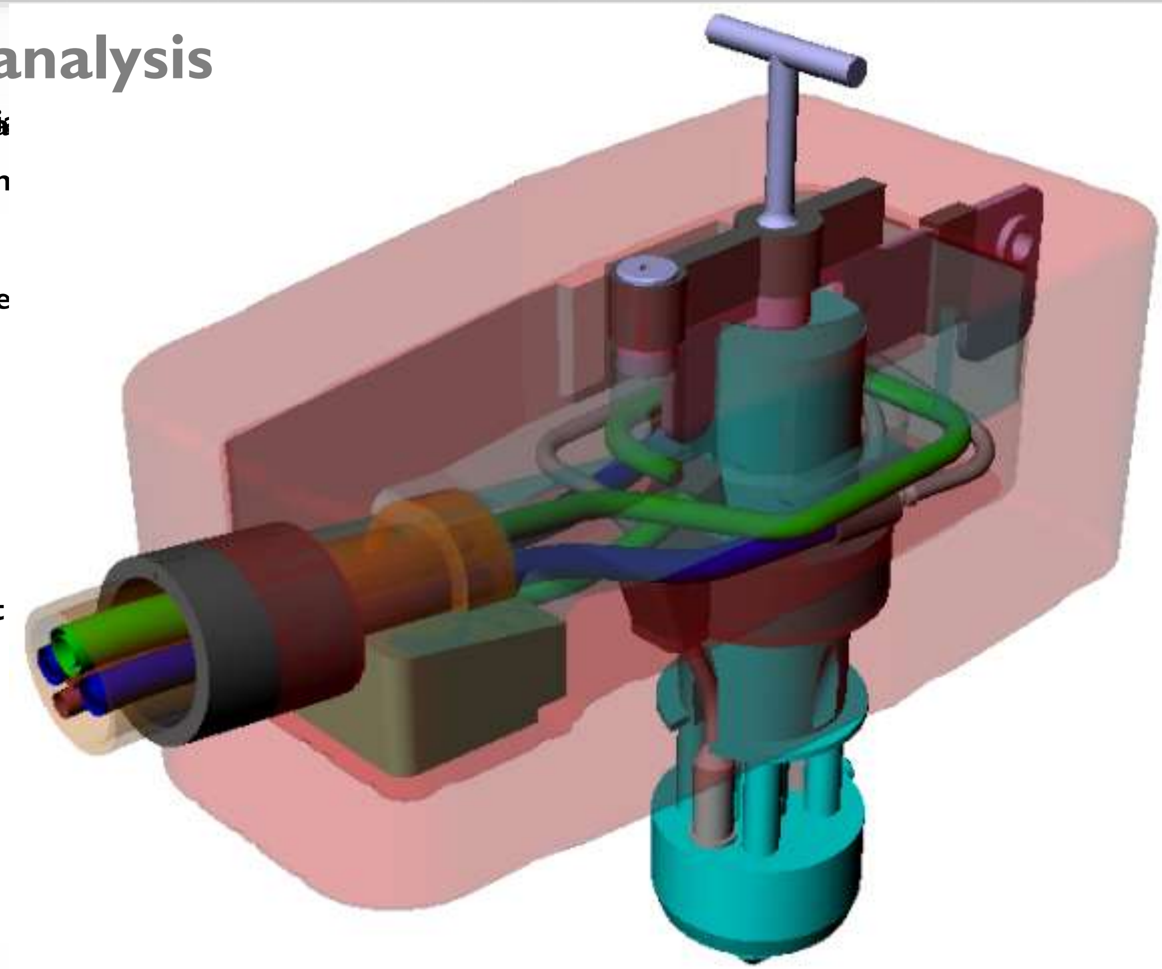
Hot stab chassis n

Carrier pipe

Material – Stainless s

Material – Nylo
with stainless s

Hot stab unit

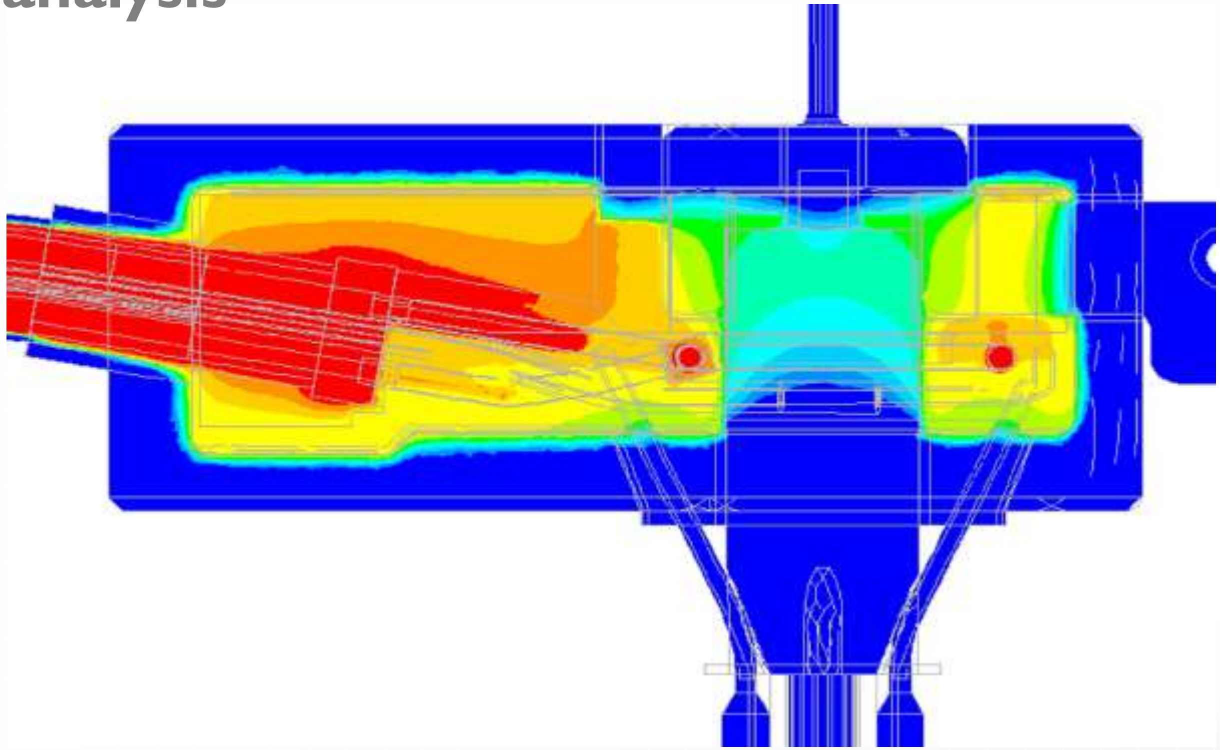


al – Stainless steel
r tube

n

Industrial application of CFD

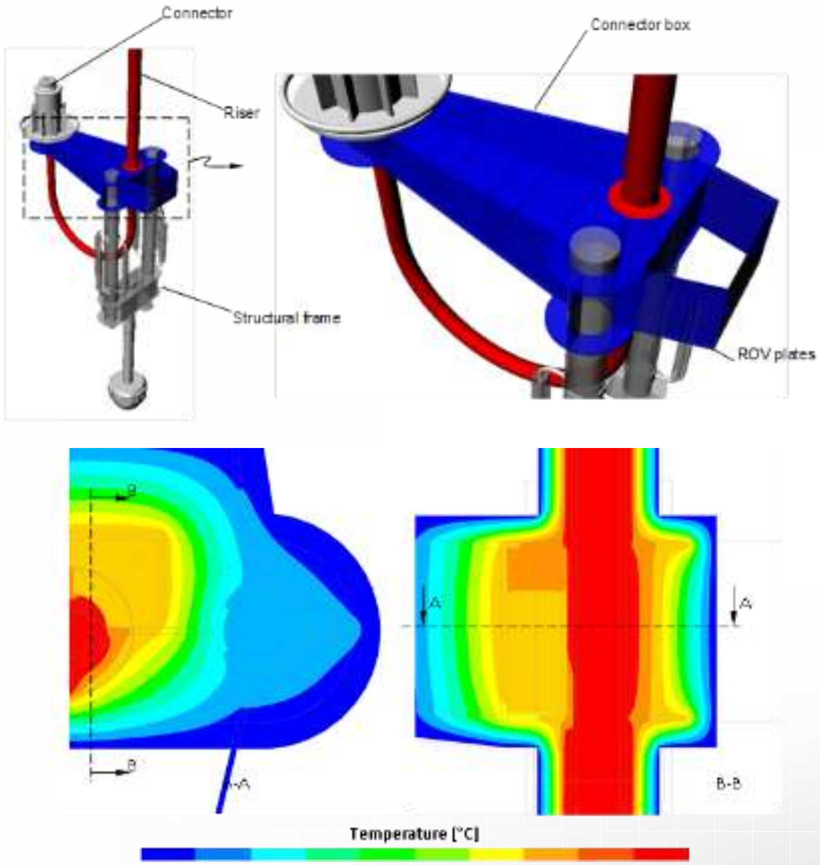
Thermal analysis



Temperature

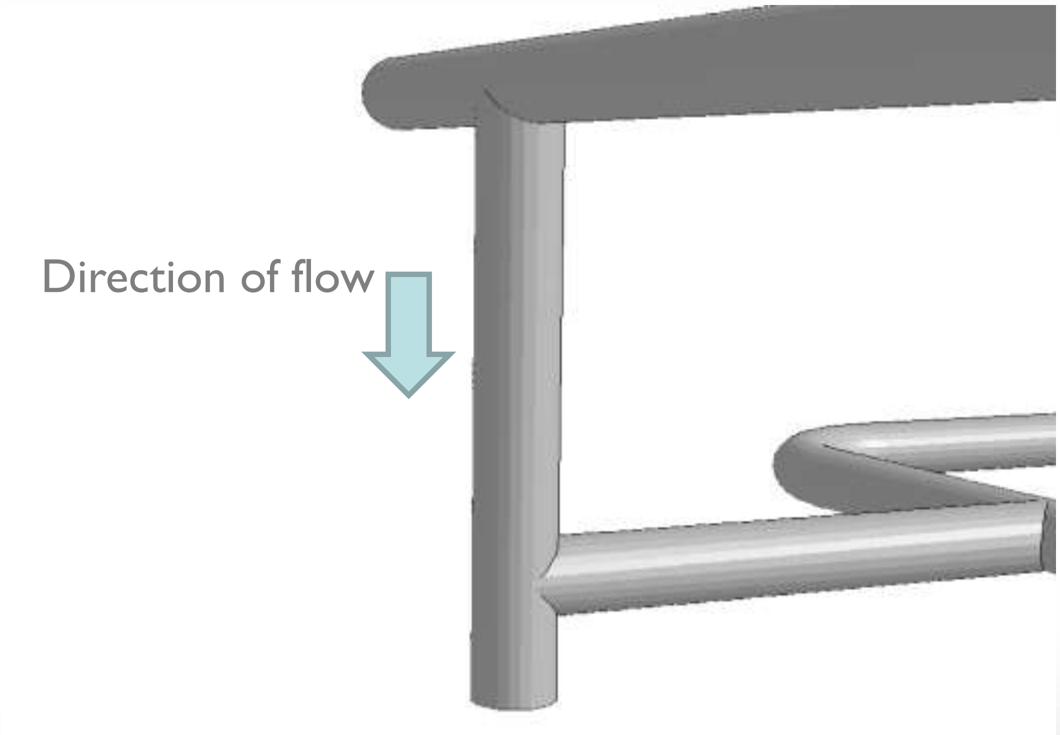
Industrial application of CFD

Thermal analysis



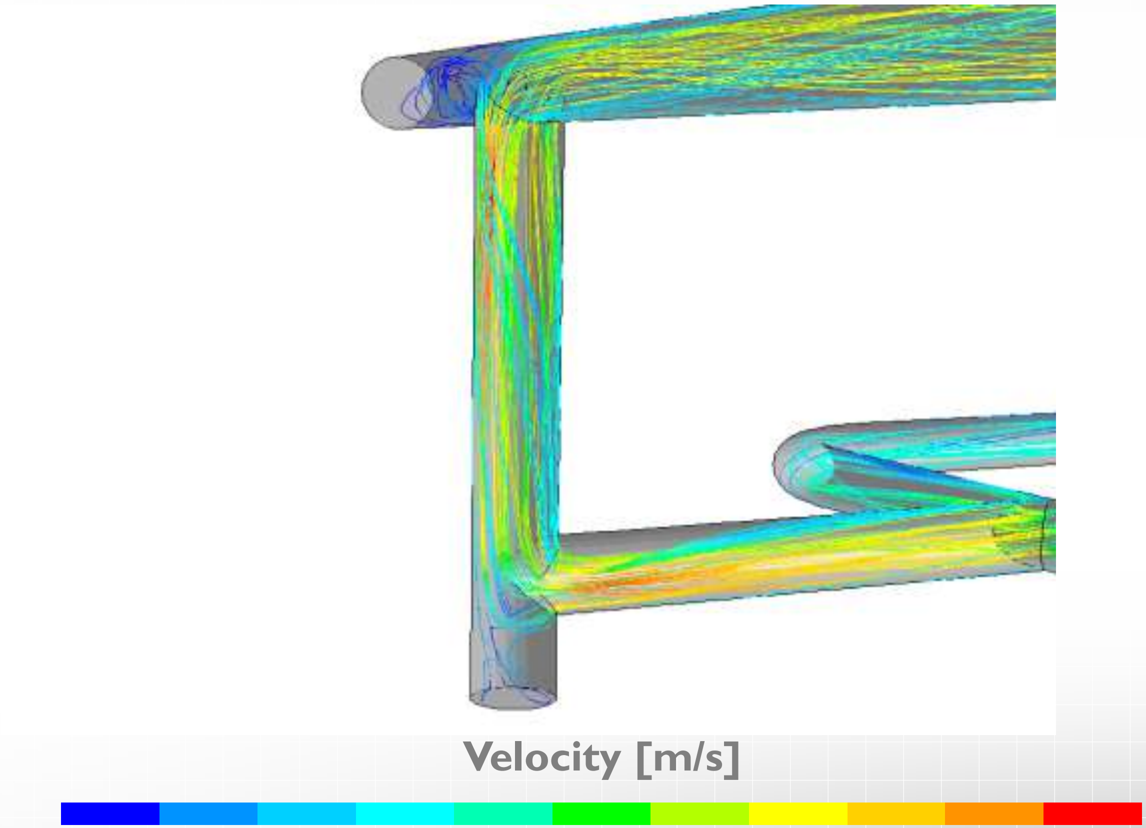
Industrial application of CFD

Erosion



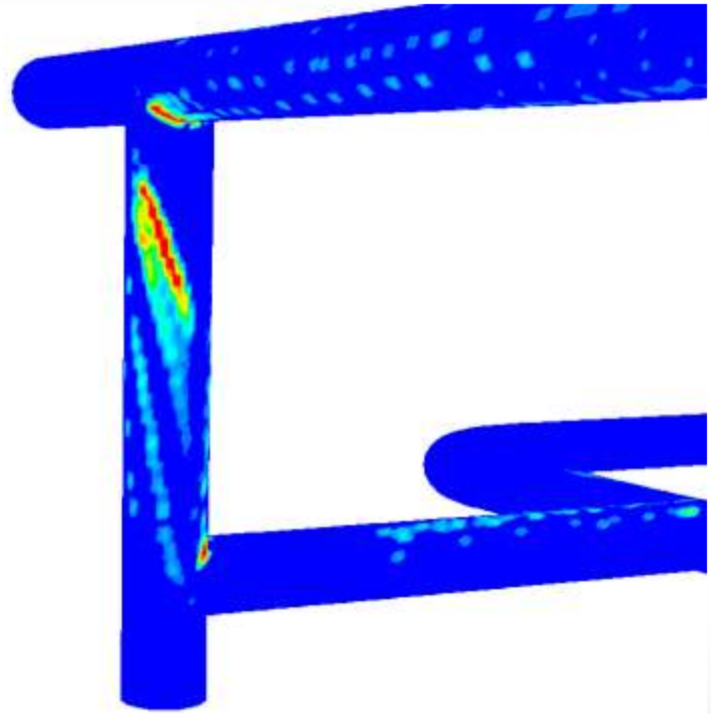
Industrial application of CFD

Erosion



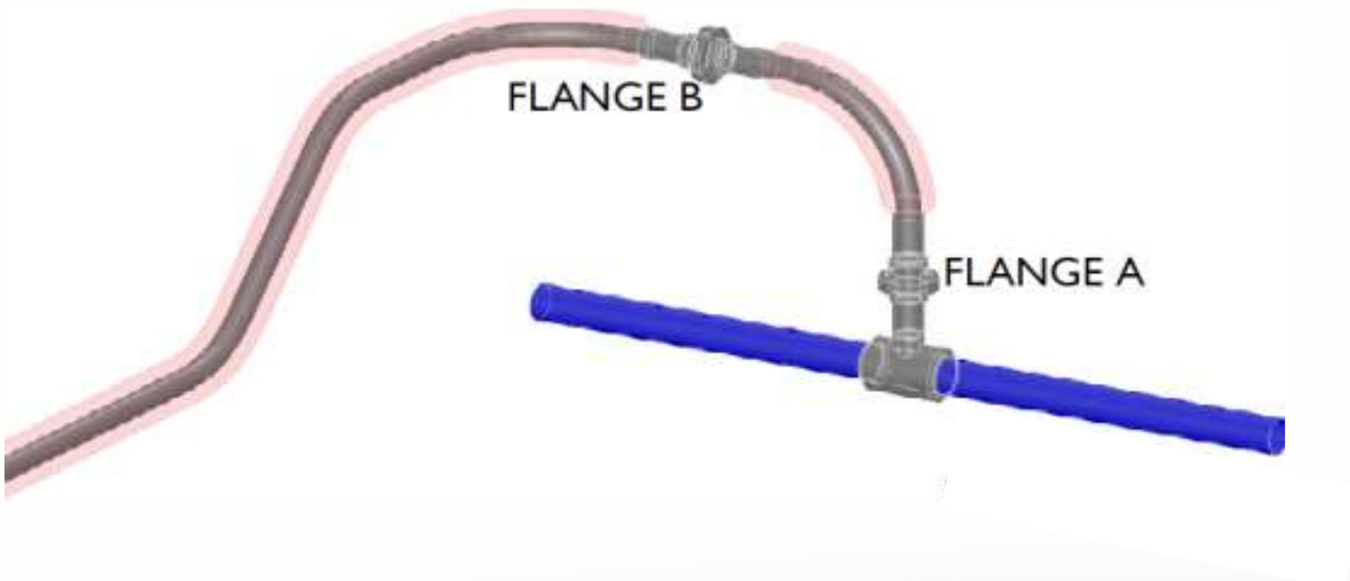
Industrial application of CFD

Erosion



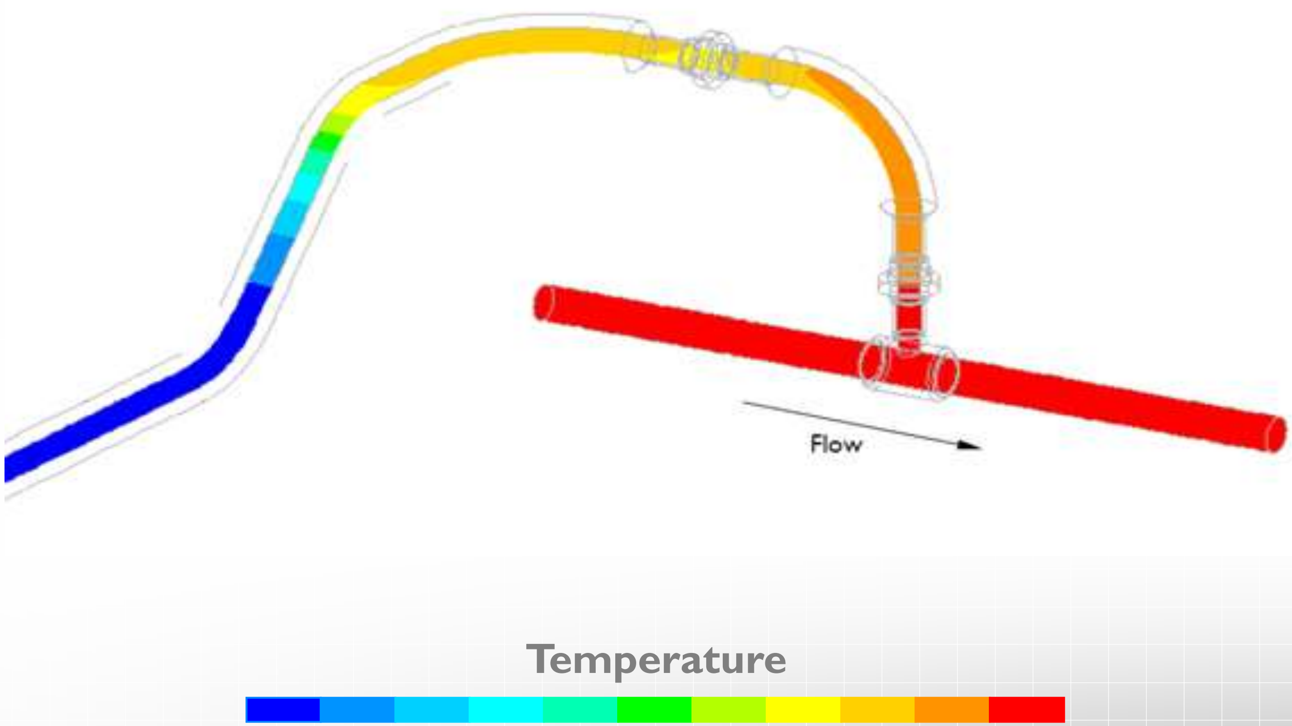
Industrial application of CFD

Condensation formation



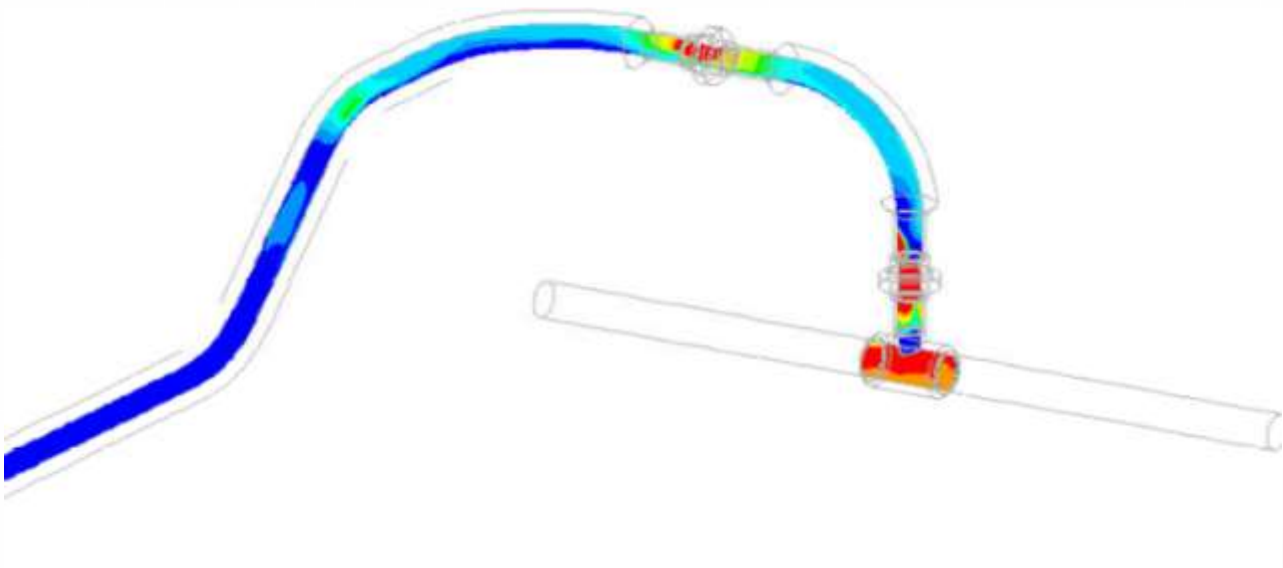
Industrial application of CFD

Condensation formation



Industrial application of CFD

Condensation formation



Rate of heat loss

Abercus Riverside House Riverside Drive Aberdeen AB11 7LH www.abercus.com

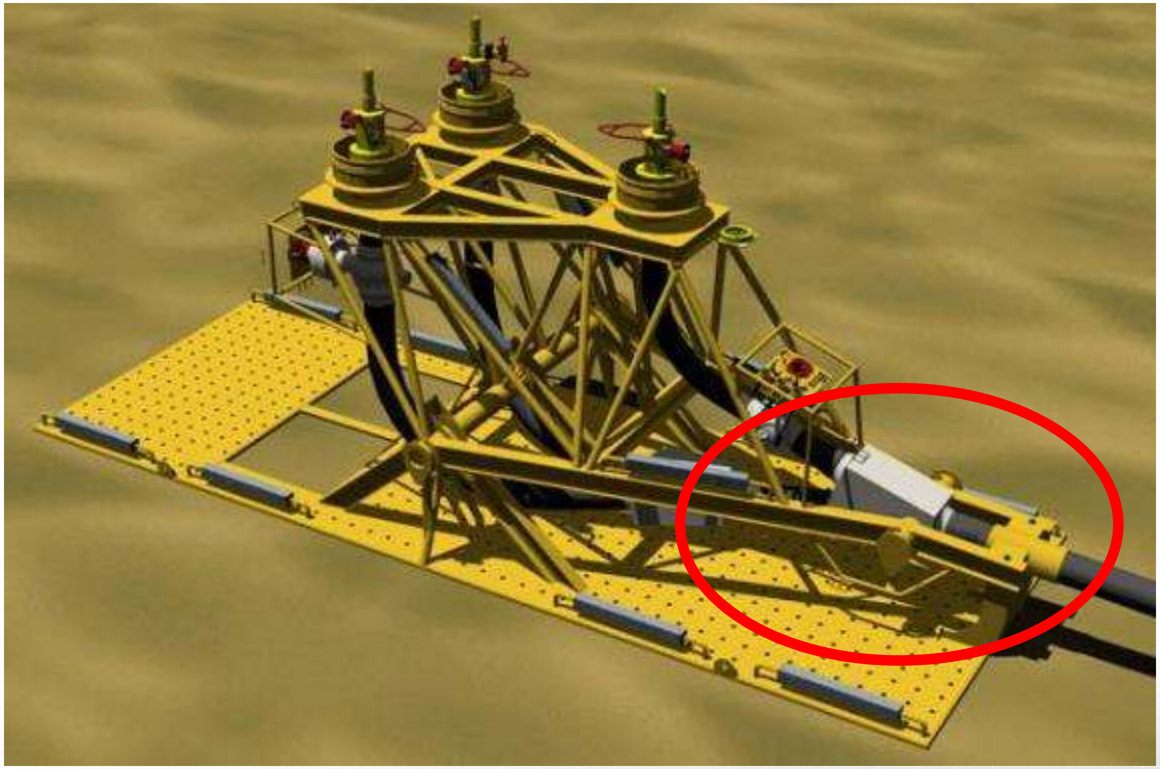
Industrial application of CFD

Sand accumulation

Sand packing density

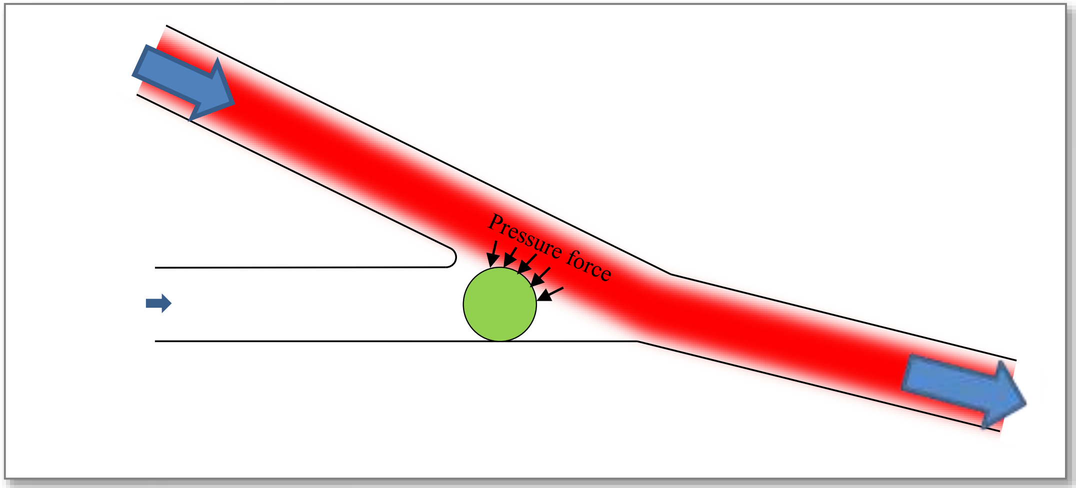
Industrial application of CFD

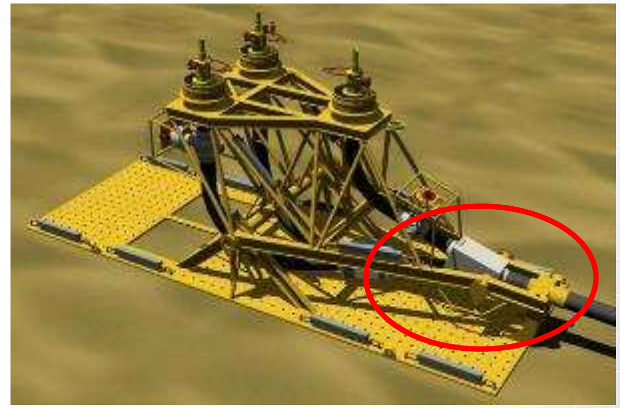
Pigging

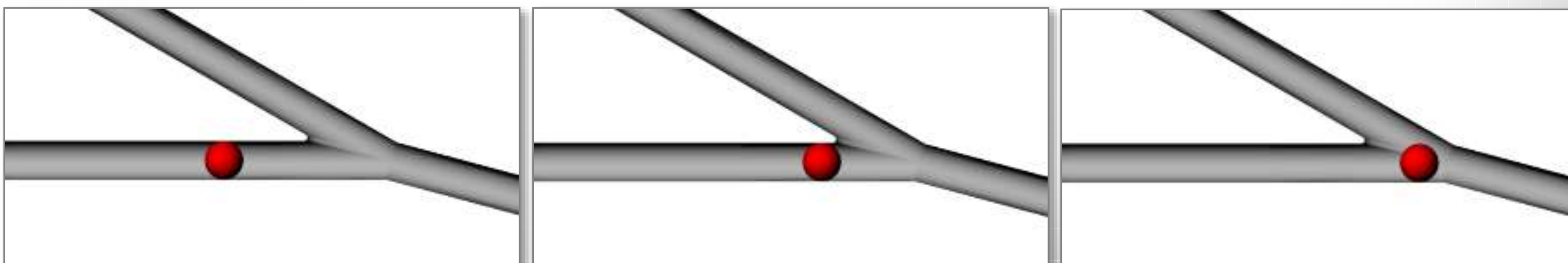


Industrial application of CFD

Pigging

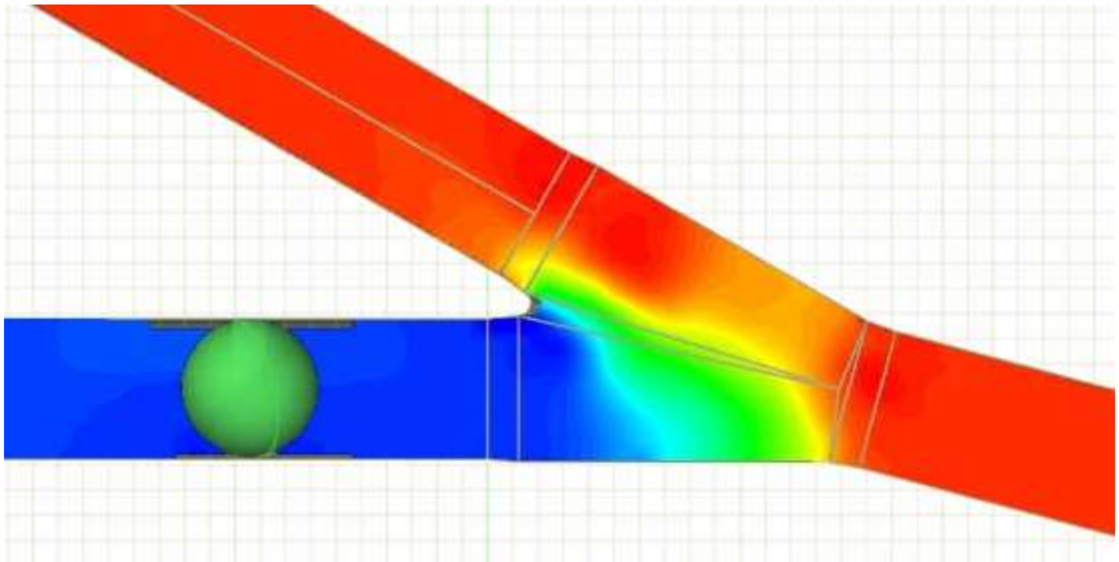


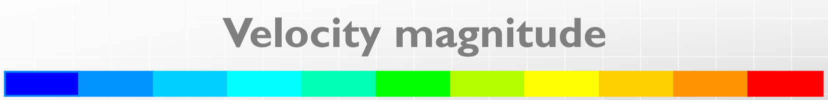




Industrial application of CFD

Pigging

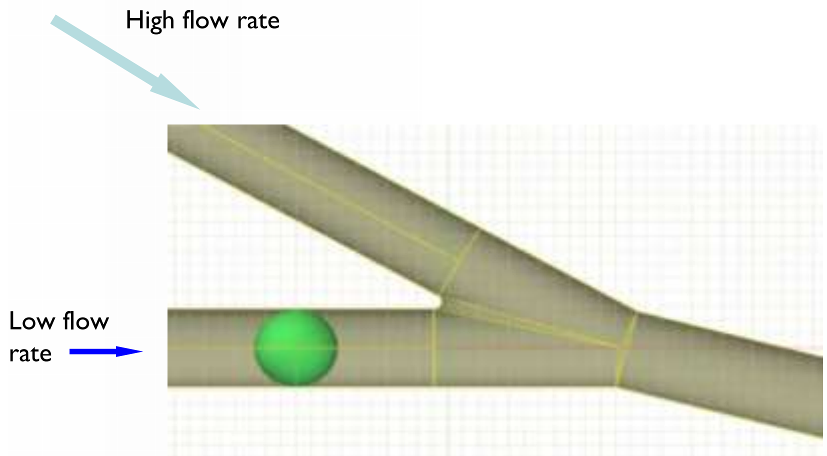




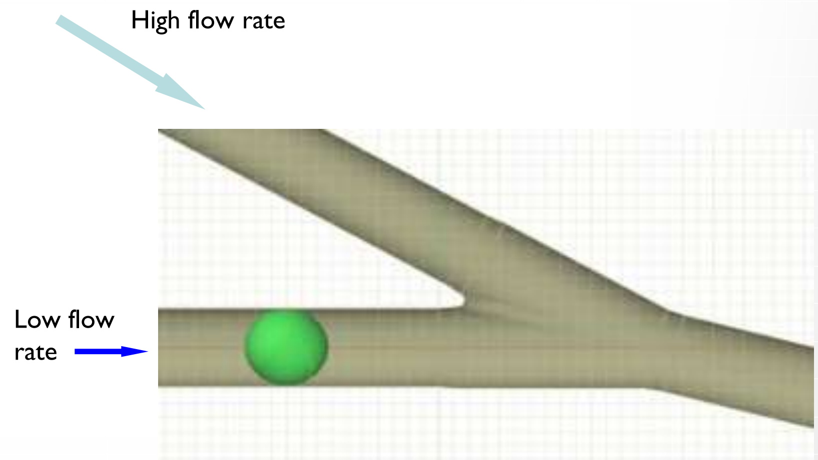
Industrial application of CFD

Pigging

Without undercut

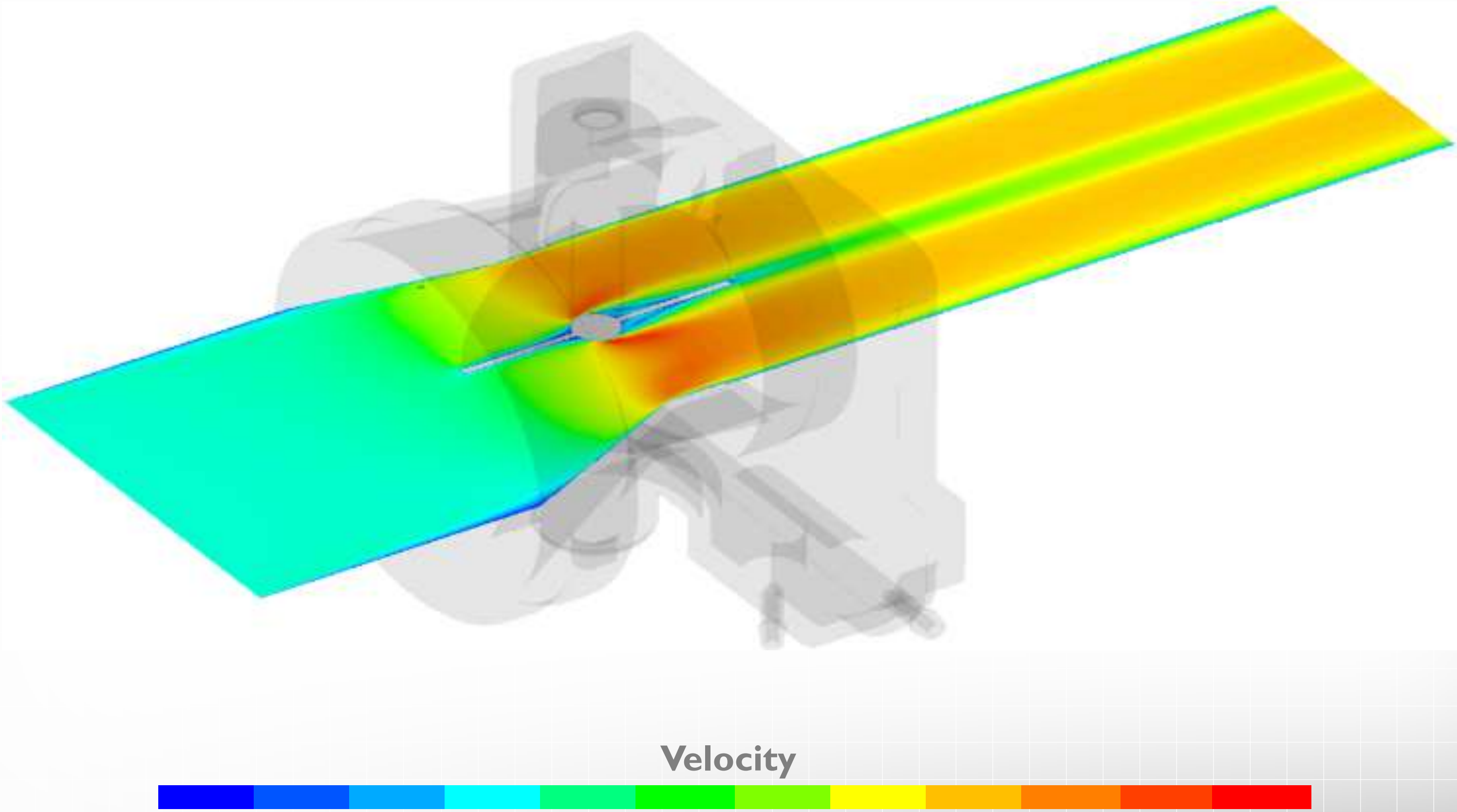


With undercut



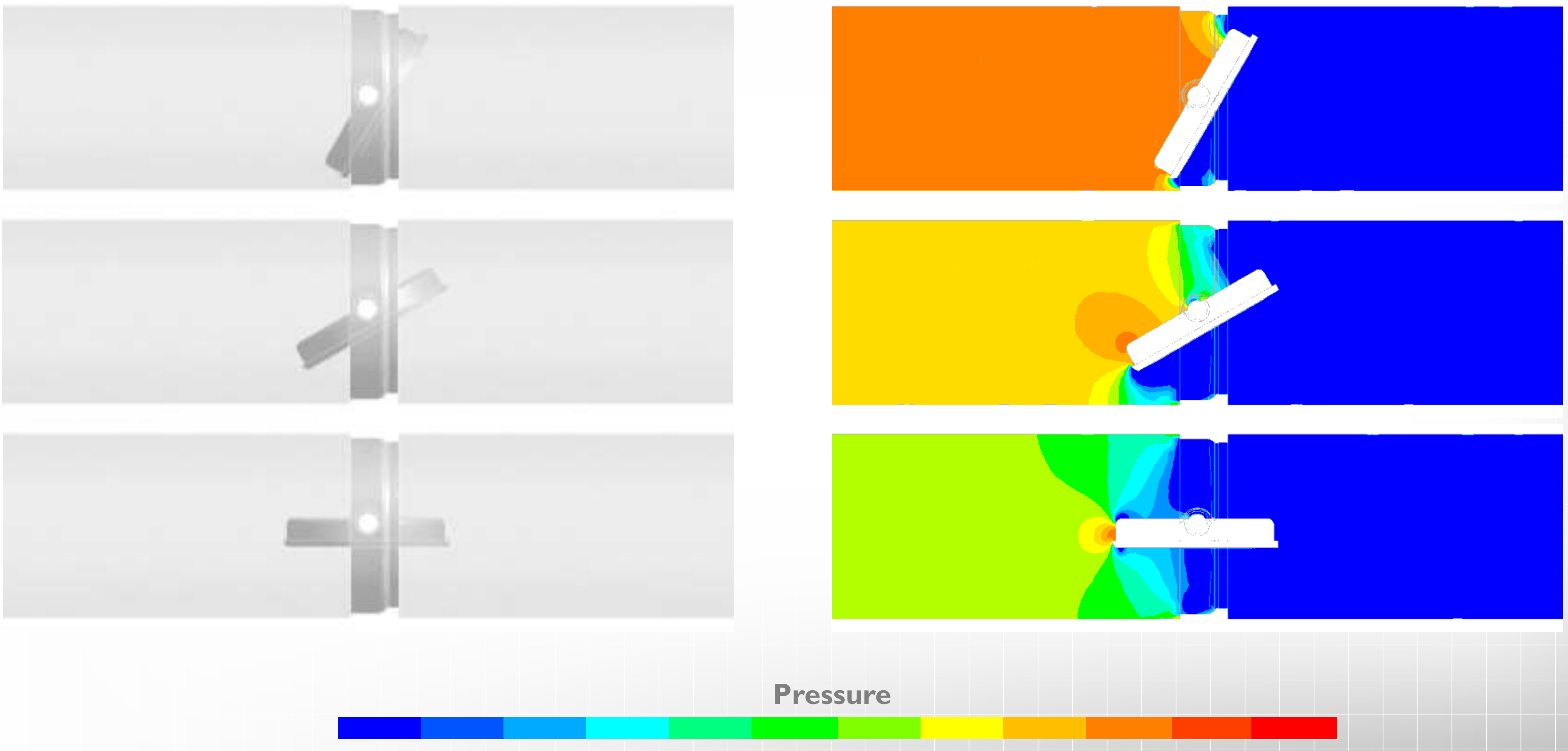
Industrial application of CFD

Valves and flow meters



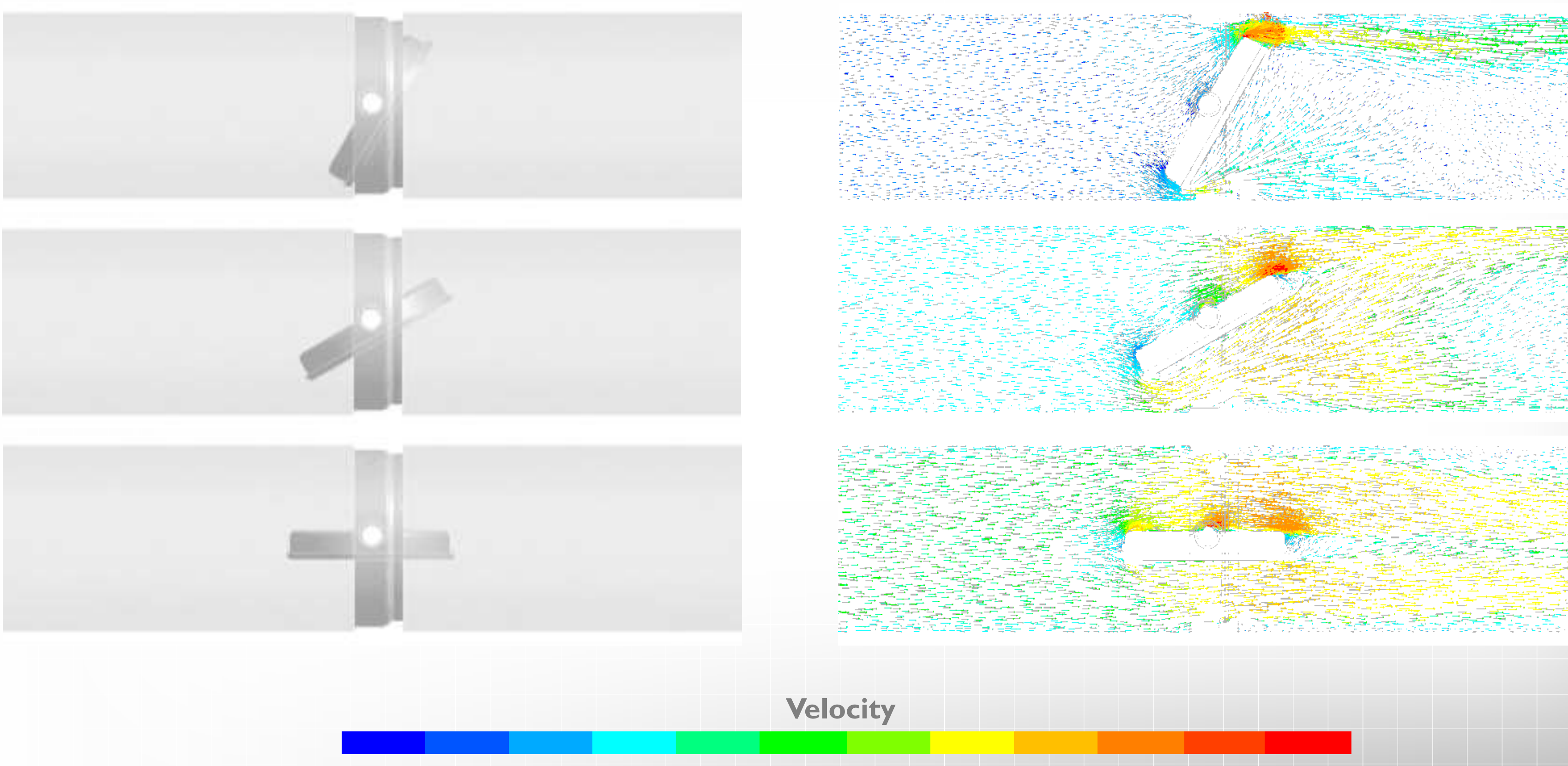
Industrial application of CFD

Valves and flow meters



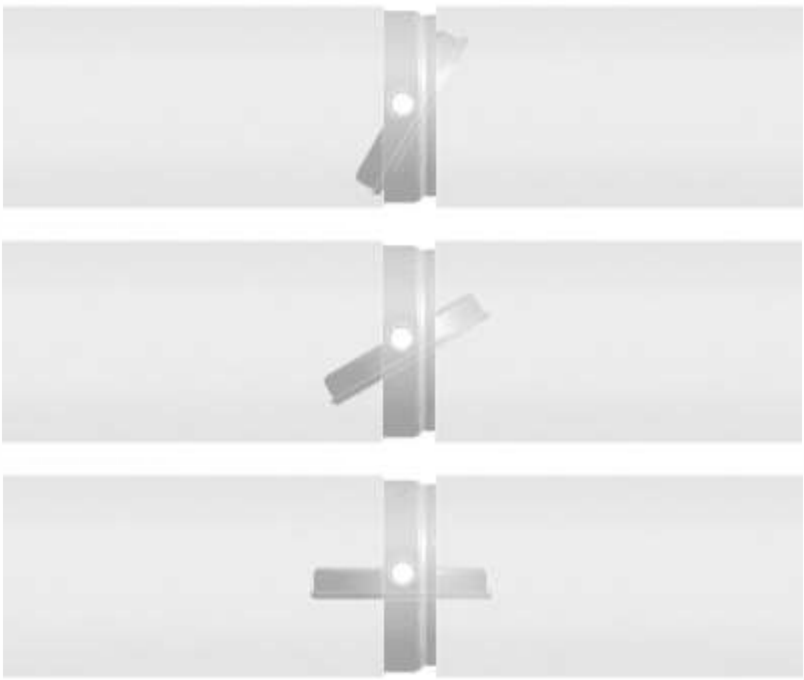
Industrial application of CFD

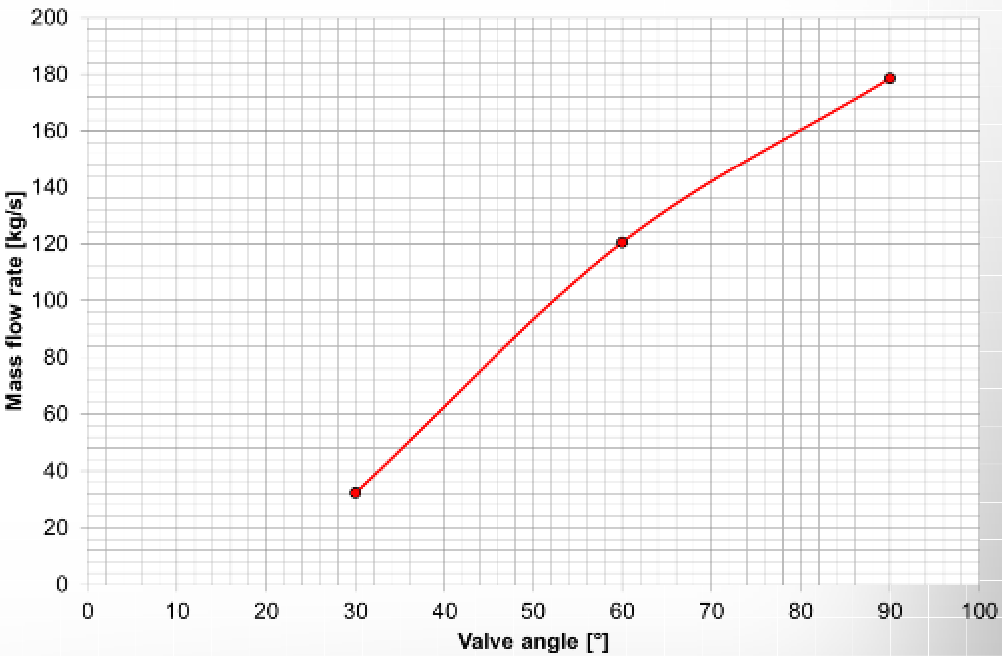
Valves and flow meters



Industrial application of CFD

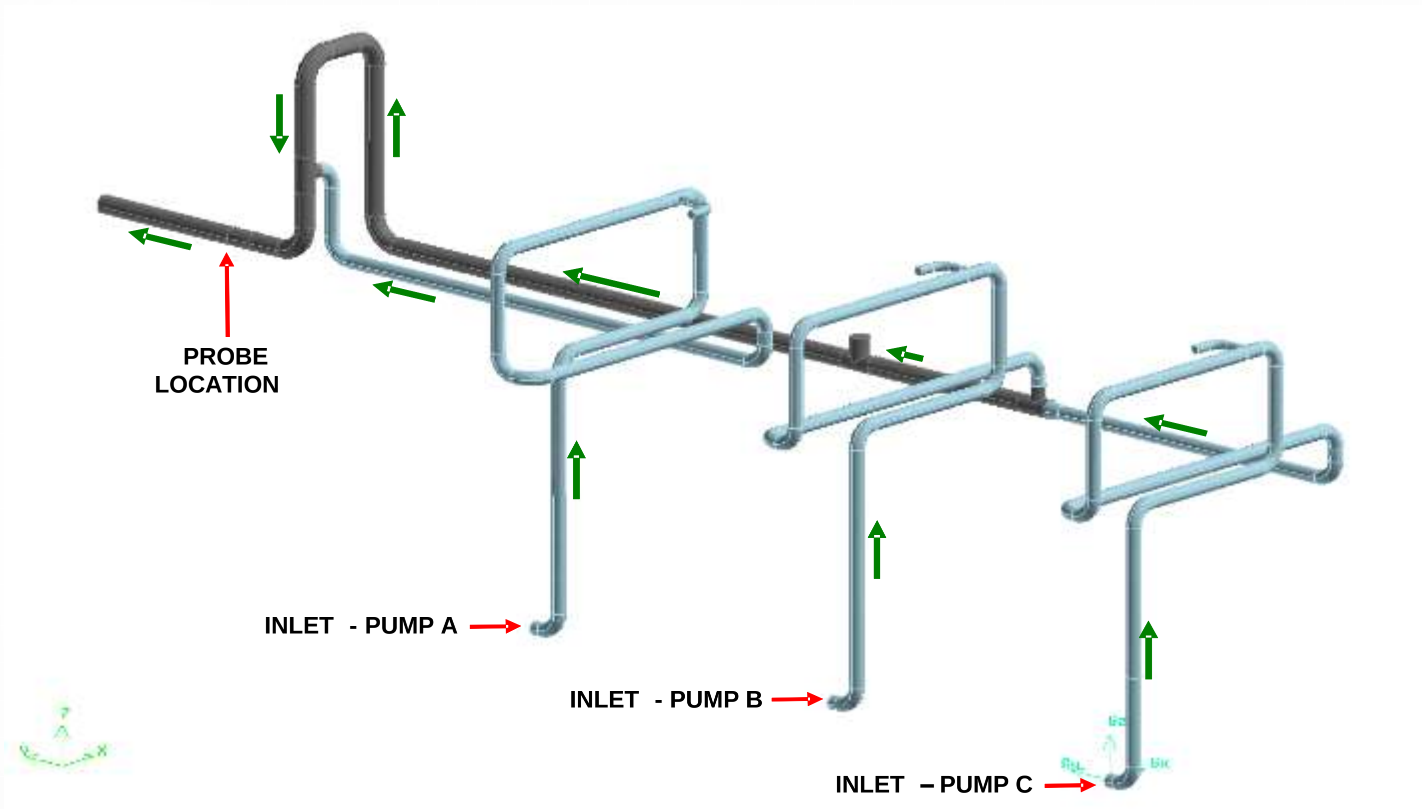
Valves and flow meters





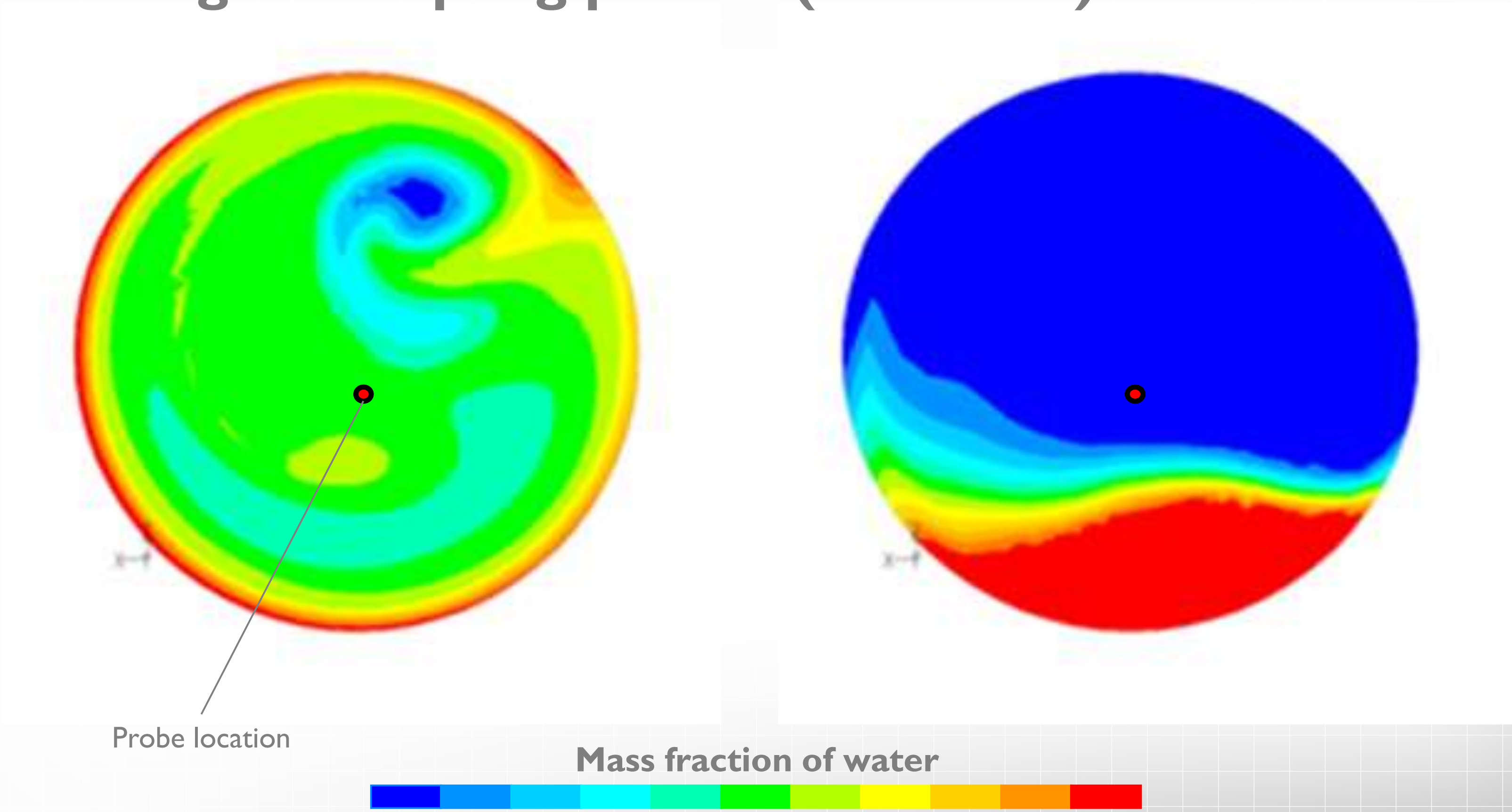
Industrial application of CFD

Positioning of sampling probes (ISO 3171)



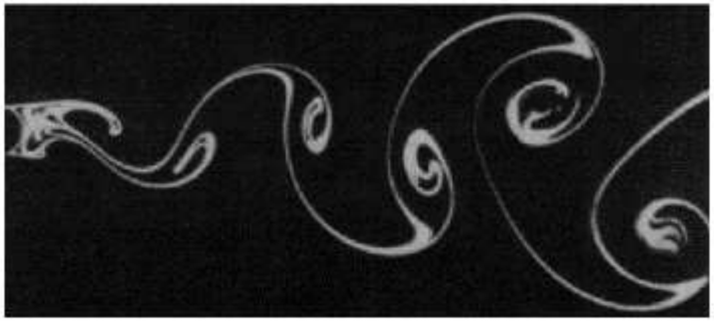
Industrial application of CFD

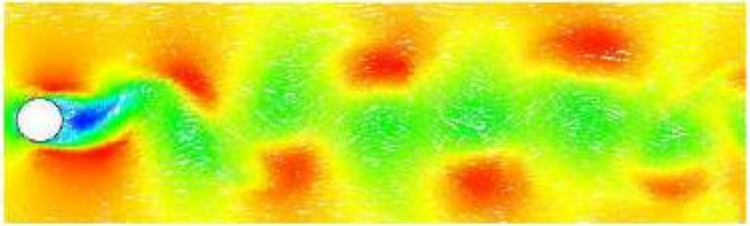
Positioning of sampling probes (ISO 3171)



Industrial application of CFD

Flow-induced vibration





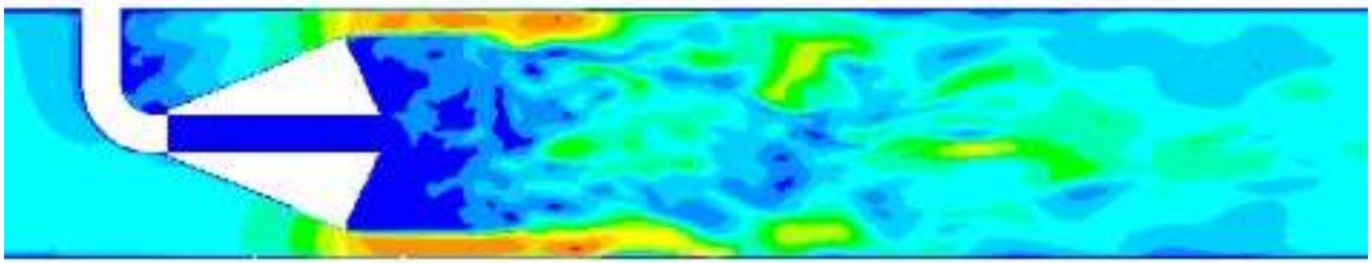
Velocity magnitude

Industrial application of CFD

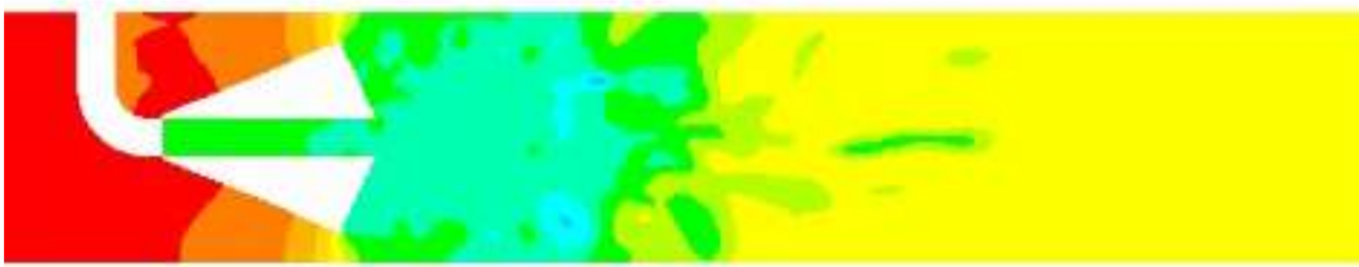
Flow-induced vibration

Industrial application of CFD

Flow-induced vibration



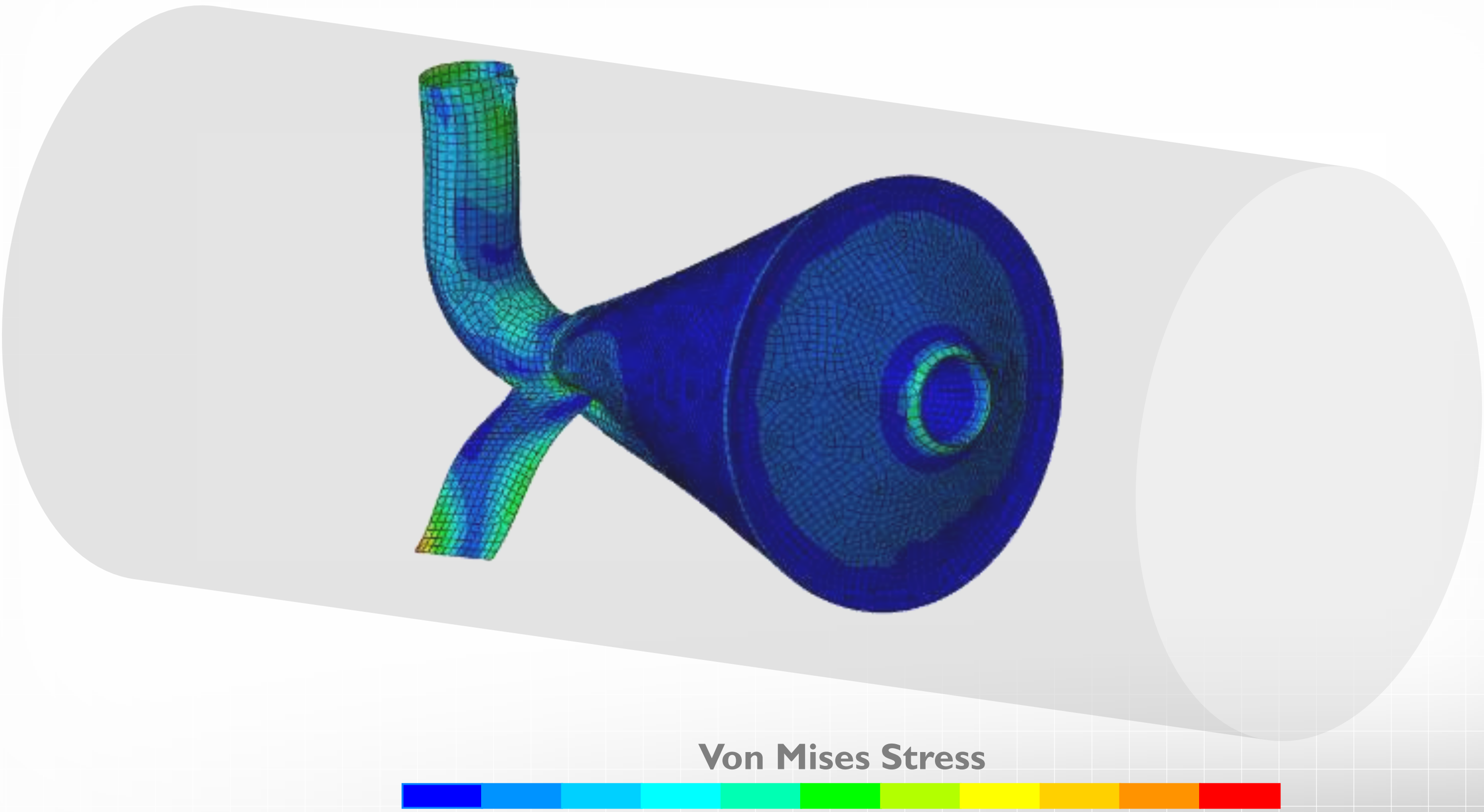
Velocity magnitude



Pressure

Industrial application of CFD

Flow-induced vibration

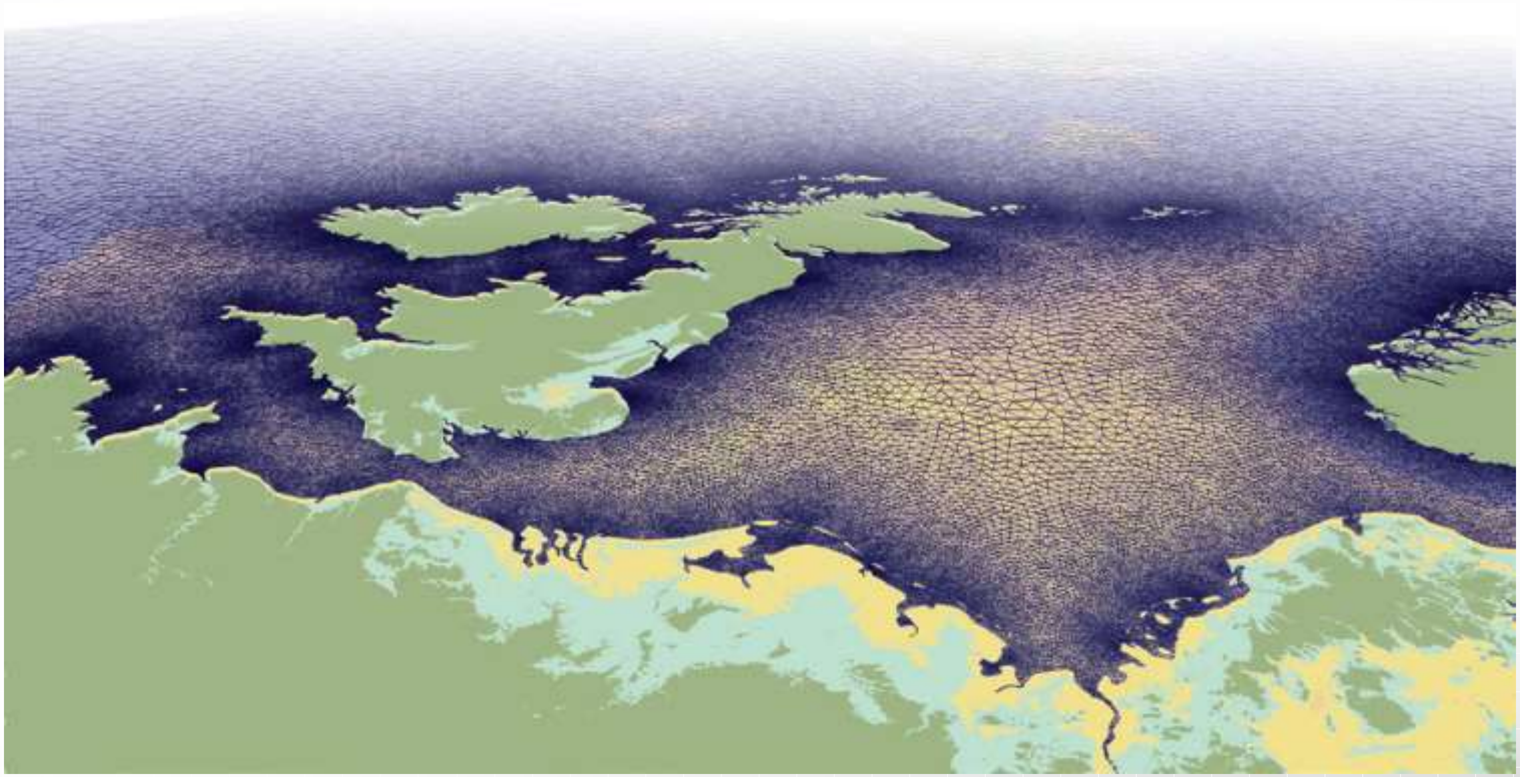


Industrial application of CFD

Flow-induced vibration

Industrial application of CFD

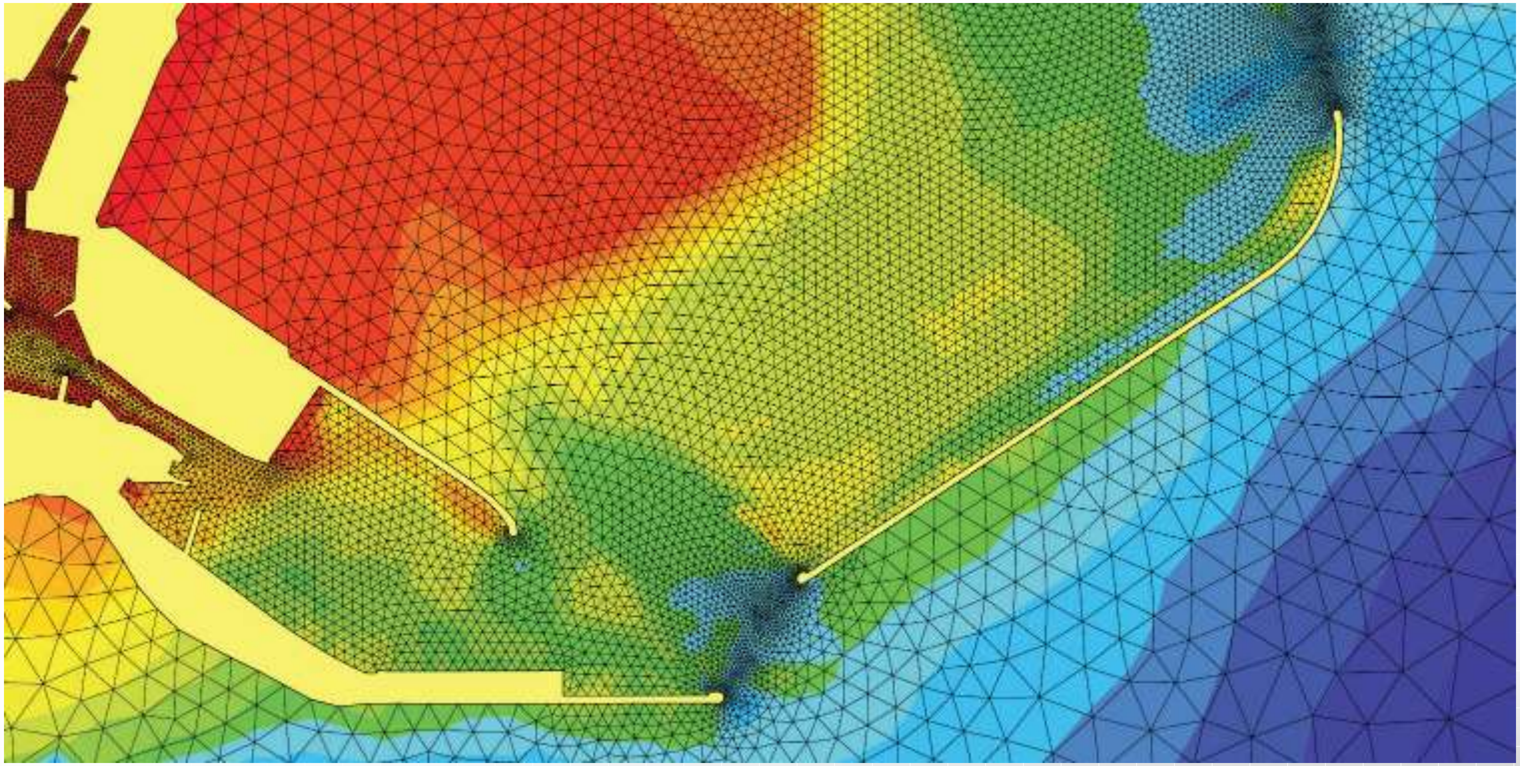
Tidal flows



Detailed continental shelf model, HR Wallingford, TELEMAC User Conference 2012.

Industrial application of CFD

Tidal flows



Detailed model of storm surge in a harbour, HR Wallingford, TELEMAC User Conference 2012.

Agenda

- Introduction
- Industrial application of CFD
- Lower cost and open source simulation tools
- Verification and validation
- Summary.

Lower cost and open source simulation tools

- Traditionally CFD and FEA tools have perhaps been considered as high cost, niche simulation tools
- The widely used general-purpose commercial codes have been developed over decades, primarily for use in other industries, and contain a huge amount of functionality that may not be used for many day-to-day applications in the subsea sector
- There is now a growing range of lower cost and open source simulation tools emerging that are accessible to everyone and are fit for purpose for many subsea applications.

Lower cost and open source simulation tools

- Abercus has developed ORTHOFLO, a structured orthogonal CFD code which is used for some niche applications and as a CFD training tool
- Abercus has also developed a suite of flow assurance tools FAST which is able to massively outperform the likes of OLGA for some basic applications but at a fraction of the cost
- Open source CFD tools include: OpenFOAM, Code_Saturne, TELEMAC, REEF3D, FEATFLOW
- Open source FEA tools include: CALCULIX, code_aster, OpenSees.

Lower cost and open source simulation tools

- It is Abercus' expectation that open source simulation tools will become increasingly used in future and this will accelerate the democratisation of advanced simulation methods
- Whilst this is a massive opportunity for our industry, we need to be rigorous with respect to **verification and validation.**

Agenda

- Introduction
- Industrial application of CFD
- Lower cost and open source simulation tools
- Verification and validation
- Summary.

Verification and validation

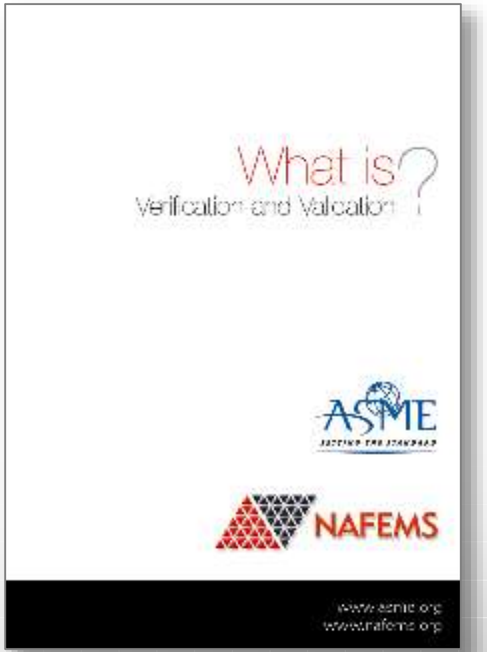
"All models are wrong but some are useful"

Robustness in the strategy of scientific model building, Box GEP,
in *Robustness in Statistics*, Launer RL and Wilkinson GN, Academic Press, pp 201–236, 1979.

- Verification and validation are the processes we must employ to gain confidence in our models, to ensure that they are useful and fit for purpose.

Verification and validation

- ASME and NAFEMS have published a *What is?* guide that is freely available for download: http://www.nafems.org/publications/browse_buy/browse_by_topic/qa/verification_and_validation/



Verification and validation

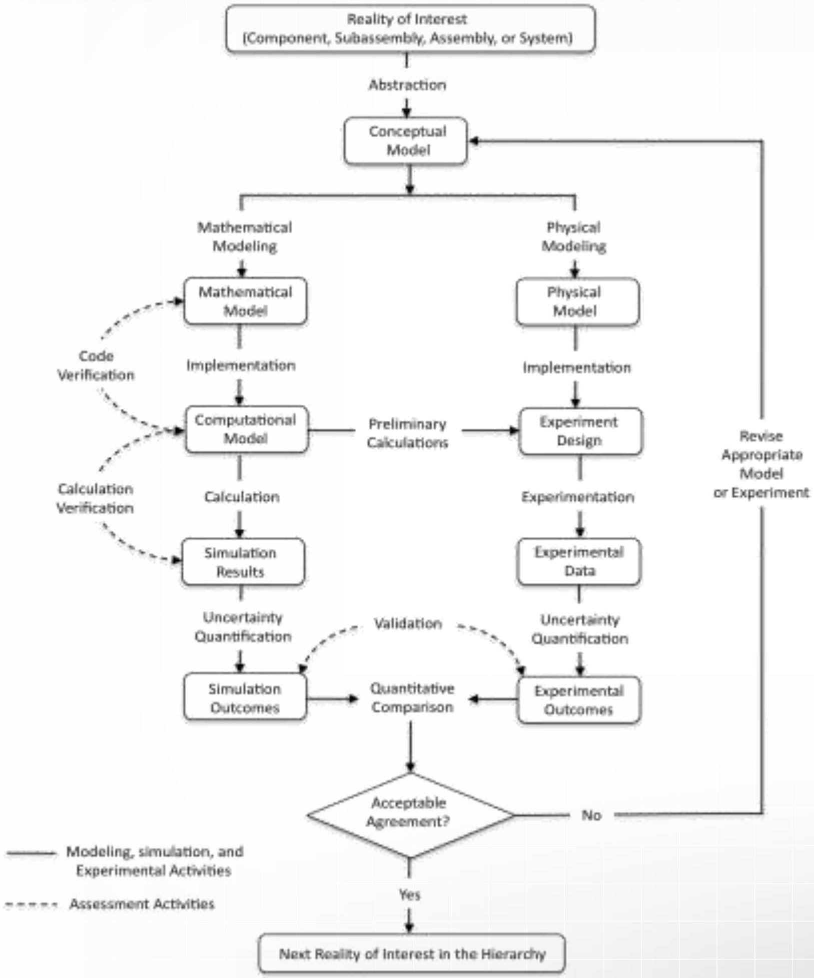
- NAFEMS is the International Association for the Engineering Modelling, Analysis and Simulation Community
- NAFEMS focuses on the practical application of numerical engineering simulation techniques such as finite element analysis, computational fluid dynamics, and multibody simulation
- There are a number of key strands to NAFEMS:
  - Teaching and training
  - PSE Scheme – to demonstrate competence
  - Verification and validation of simulation methods
  - National/international conferences to promote exchange of ideas
- http://www.nafems.org/.

Verification and validation

- **Verification**: the process of determining that a computational model accurately represents the underlying mathematical model and its solution
- **Validation**: the process of determining the degree to which a model is an accurate representation of the real world from the perspective of the intended uses of the model

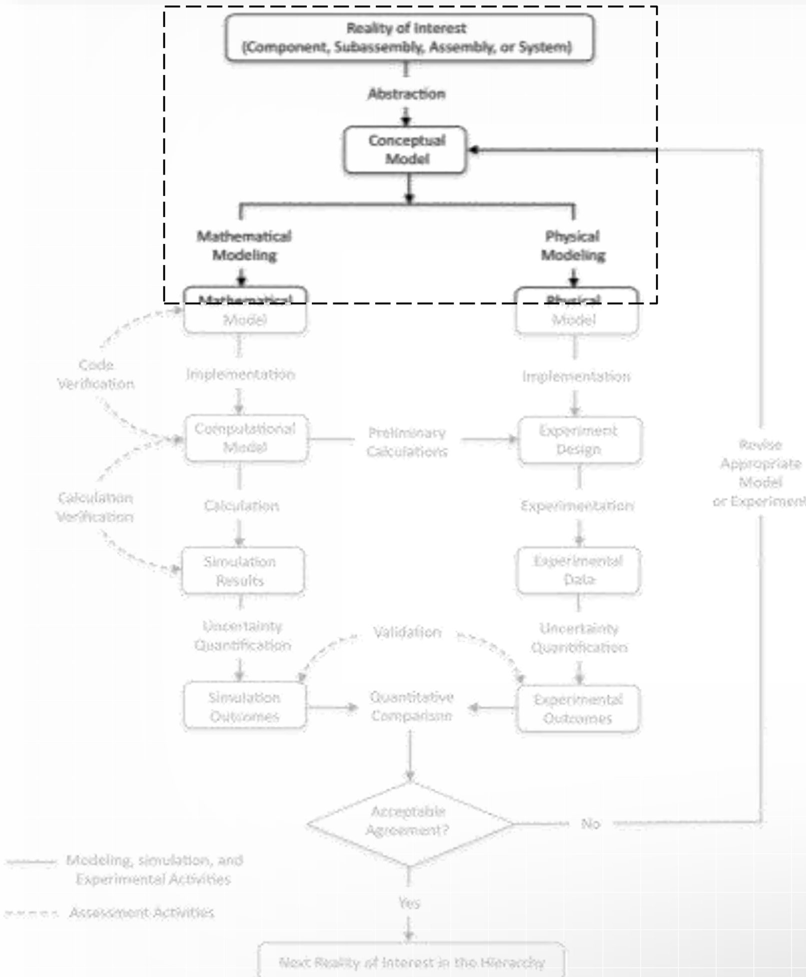
- Verification is the domain of mathematics and validation is the domain of physics.

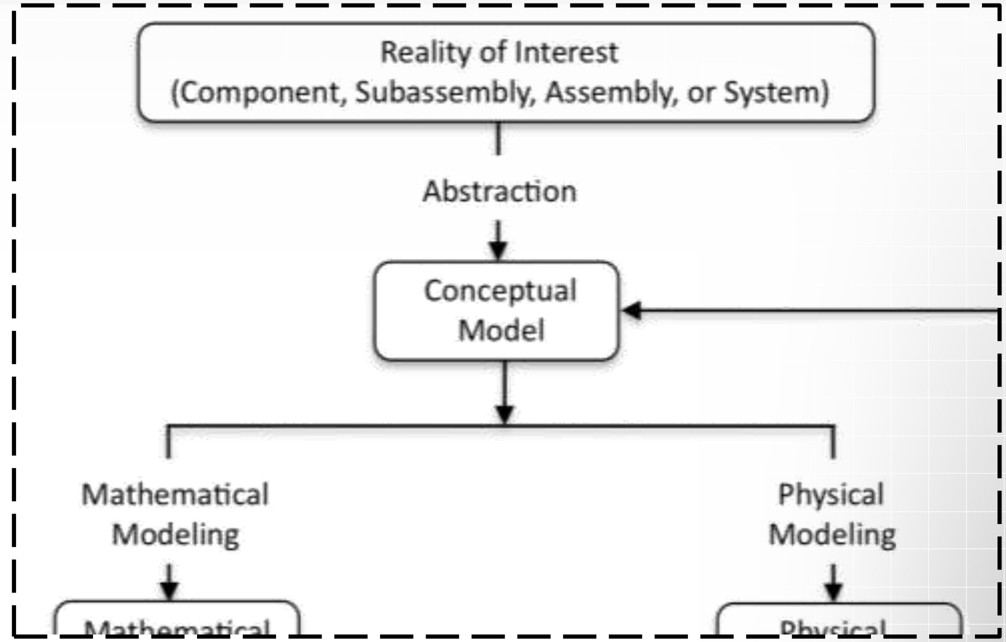
Verification and validation



From ASME/NAFEMS *What is?* Guide.

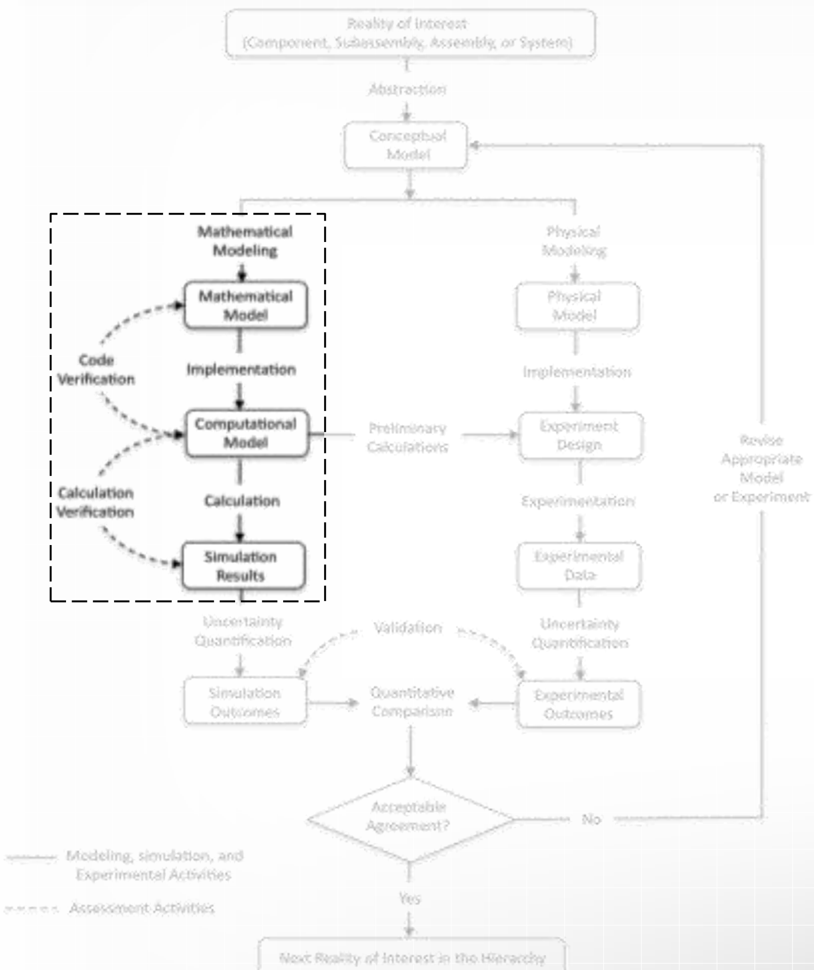
Verification and validation

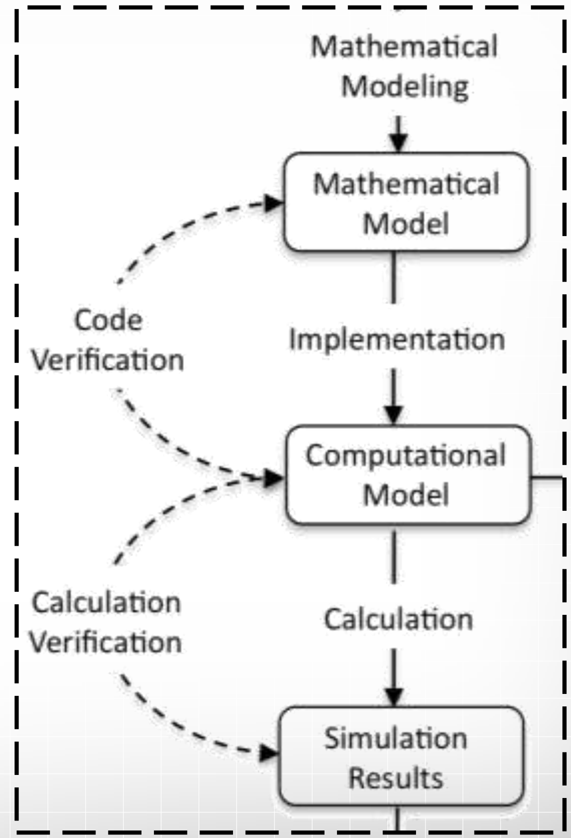




From ASME/NAFEMS *What is?* Guide.

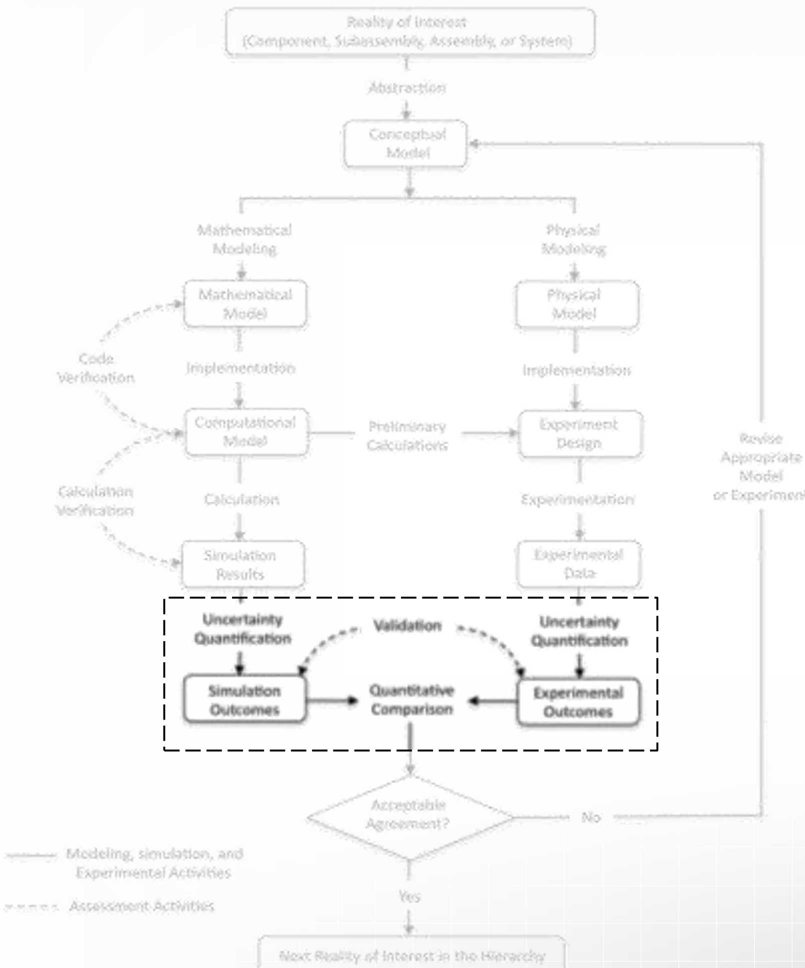
Verification and validation

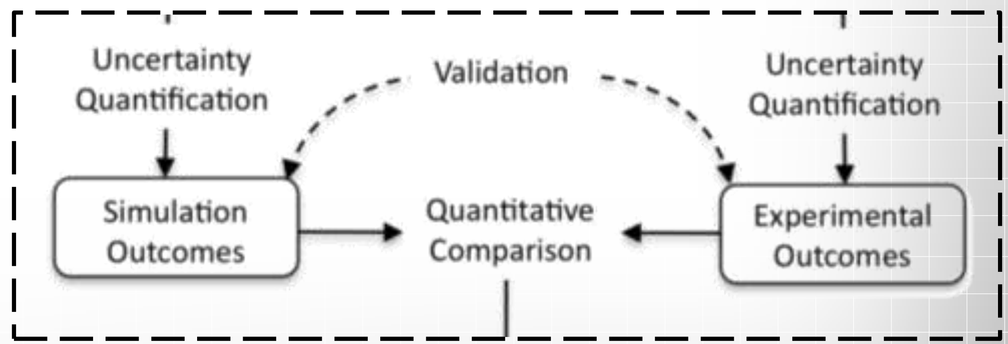




From ASME/NAFEMS *What is?* Guide.

Verification and validation





From ASME/NAFEMS *What is?* Guide.

Verification and validation

- One of the major benefits of CFD and FEA is that they are *first principles* approaches, which enables a large degree of flexibility on the applications to which it can be applied
- However… with this flexibility come great responsibility
- CFD and FEA can be misused
- The abstraction and derivation of the mathematical model is entirely down to the analyst/engineer
- The issue of verification and validation is hugely important for gaining confidence in the CFD and FEA approaches.

Verification and validation

- Often, the issue is not whether CFD or FEA can model something – it's the validation of the approach for the application of interest which is crucial
- It's important to recognise the envelope of applicability for the tools used and choose an appropriate **fit for purpose** tool for the application of interest
- Do not blame CFD and FEA tools if they don't yield a useful prediction
  - They are verified for solving equations, so if they yield dubious predictions it's probable that the conceptual model has not been correctly defined, or the simulation workflow has not been verified by the analyst.

Verification and validation

- Benchmark data is incredibly important for the purpose of validation activities and there is always a need for more reliable benchmark data, particularly for subsea engineering
- There are some repositories of benchmark data to be aware of:
  - NAFEMS (http://www.nafems.org/)
  - ERCOFTAC (http://www.ercoftac.org/)
  - QNET (http://uriah.dedi.melbourne.co.uk/w/index.php/Main_Page)
  - MARNET (https://pronet.atkinsglobal.com/marnet/)
  - CFD-online (http://www.cfd-online.com/Wiki/Main_Page)
- **We need more public sources of benchmark data.**

Agenda

- Introduction
- Industrial application of CFD
- Lower cost and open source simulation tools
- Verification and validation
- Summary.

Summary

- CFD and FEA are powerful tools that are increasingly used:
  - to deliver valuable insight at the design stage
  - to provide improved understanding of installation and operational issues
  - to demonstrate technology readiness for novel products and approaches
- Just be mindful that CFD and FEA may not always be appropriate – if simpler methods are **fit for purpose**, use them!

- Benefits – first principles, the general transport equation
- Limitations – numerical diffusion
- Verification and validation – NAFEMS and its PSE scheme.

Summary

- Traditionally CFD and FEA tools have perhaps been considered as high-cost niche simulation tools
- There is now a growing range of lower cost and open source fit for purpose simulation tools emerging that can be successfully employed within industry
- It is Abercus' expectation that open source simulation tools will become increasingly used in future and this will accelerate the democratisation of advanced simulation methods
- Whilst this is a massive opportunity for our industry, we need to be rigorous with respect to **verification and validation**.