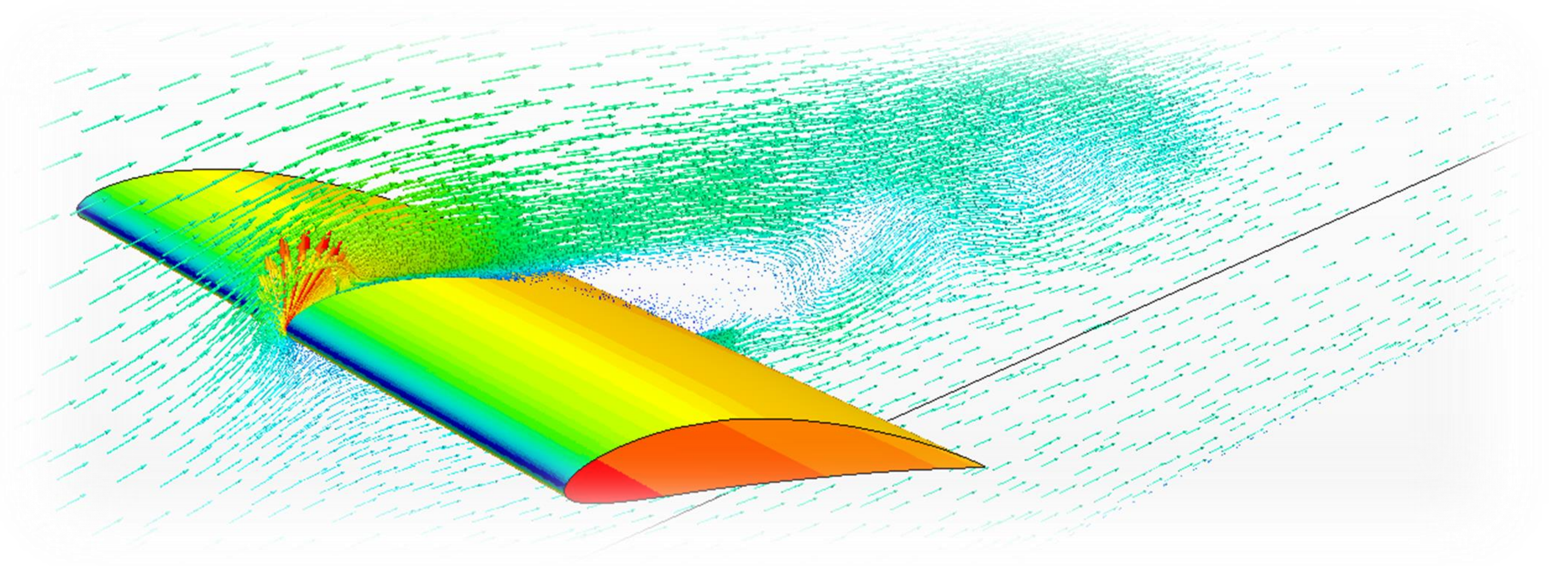
Computational Fluid Dynamics Fast Start

Laurence Marks

November 2023



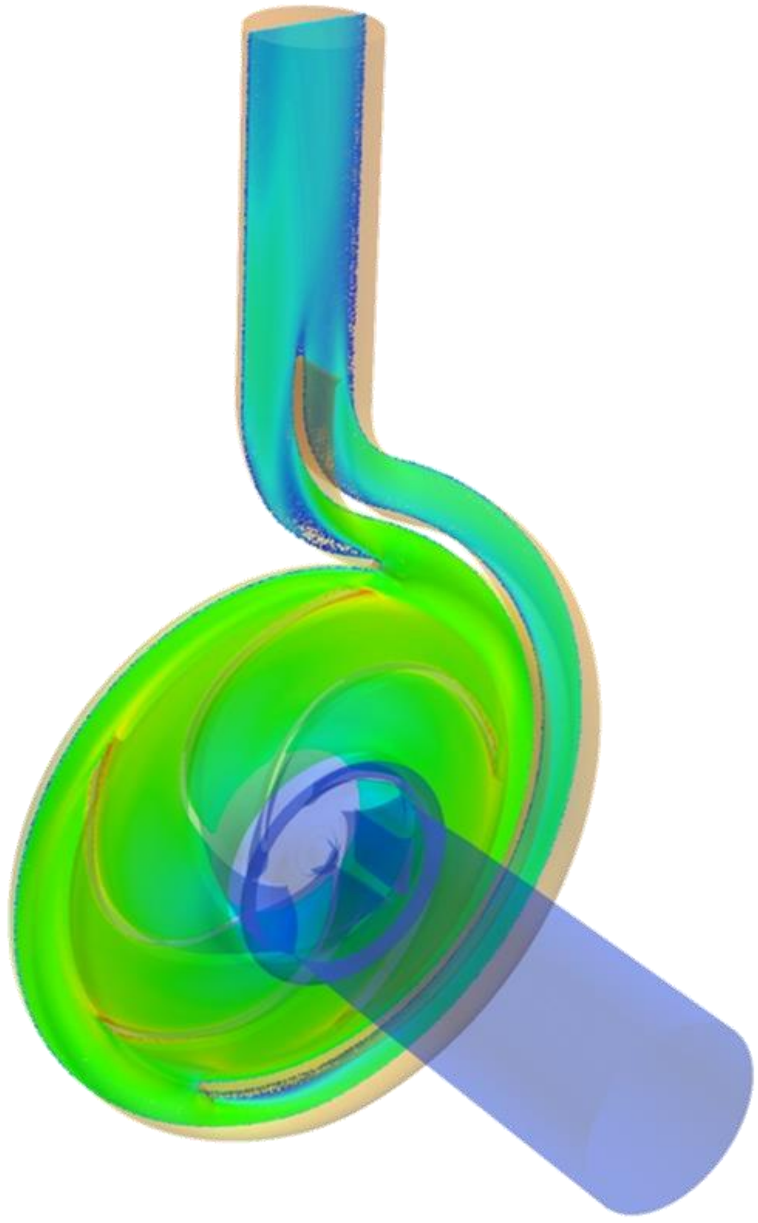
OXFORD BROOKES UNIVERSITY

Laurence Marks
07307 588760
laurencemarks64@googlemail.com

What we are doing here?

- Getting up to speed as fast as possible with the basics of <u>doing</u> computational fluid dynamics studies
  - Intro lecture
  - Demonstration
  - Tutorial

- If we get over the initial steep learning curve we can get on and use this stuff properly.
- And we'll leave the theory for another time, but don't underestimate it's importance.

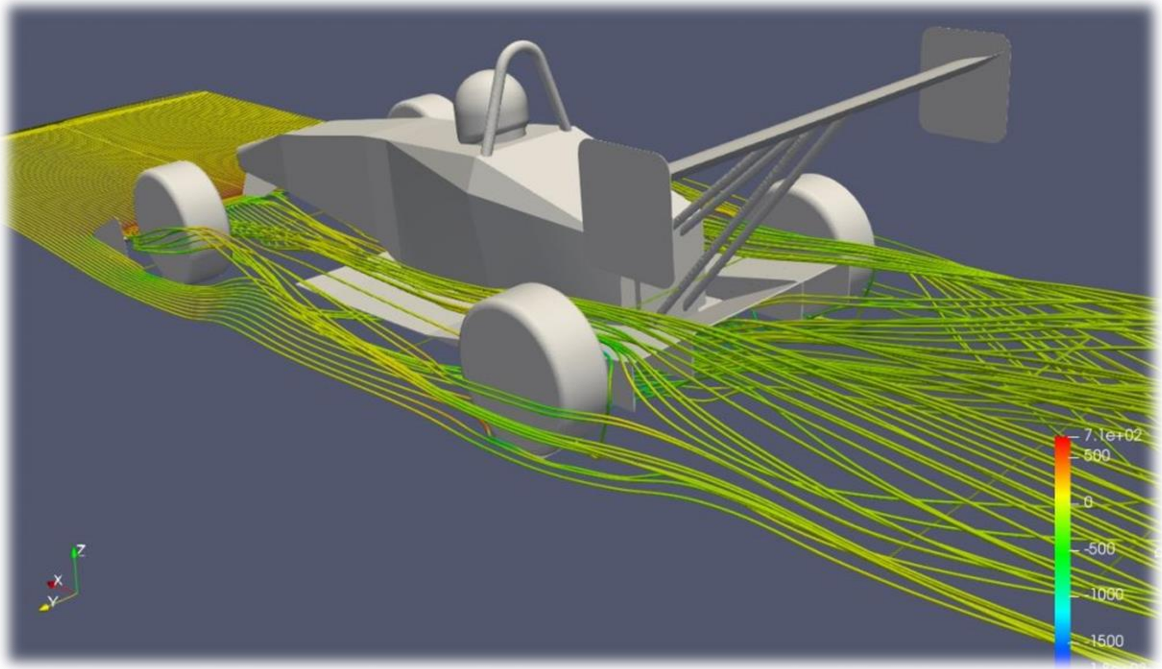


OXFORD BROOKES UNIVERSITY

Laurence Marks
07307 588760
laurencemarks64@googlemail.com

Course Contents

- What is CFD?
- What sort of things are studied using CFD?
- What's in a CFD model?
- Meshes
- Boundary layers
- Turbulence
- Geometry models for CFD
- Solving
- Results
- Process – the workflow
- What do we do with the results…

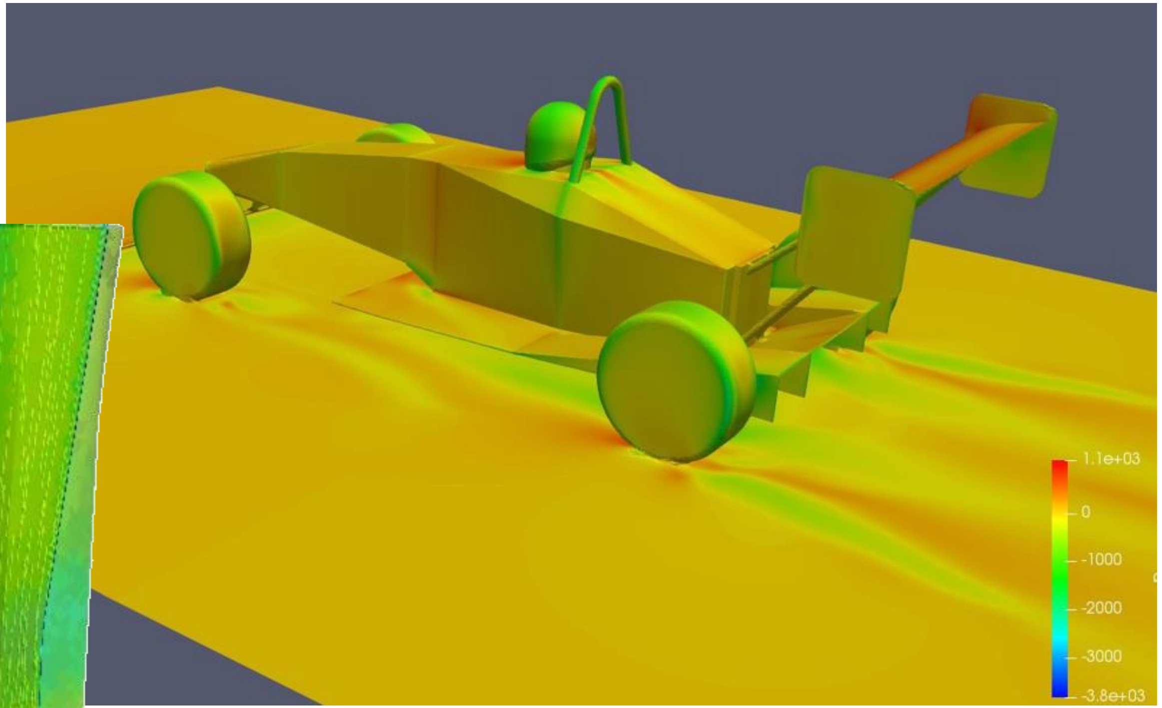


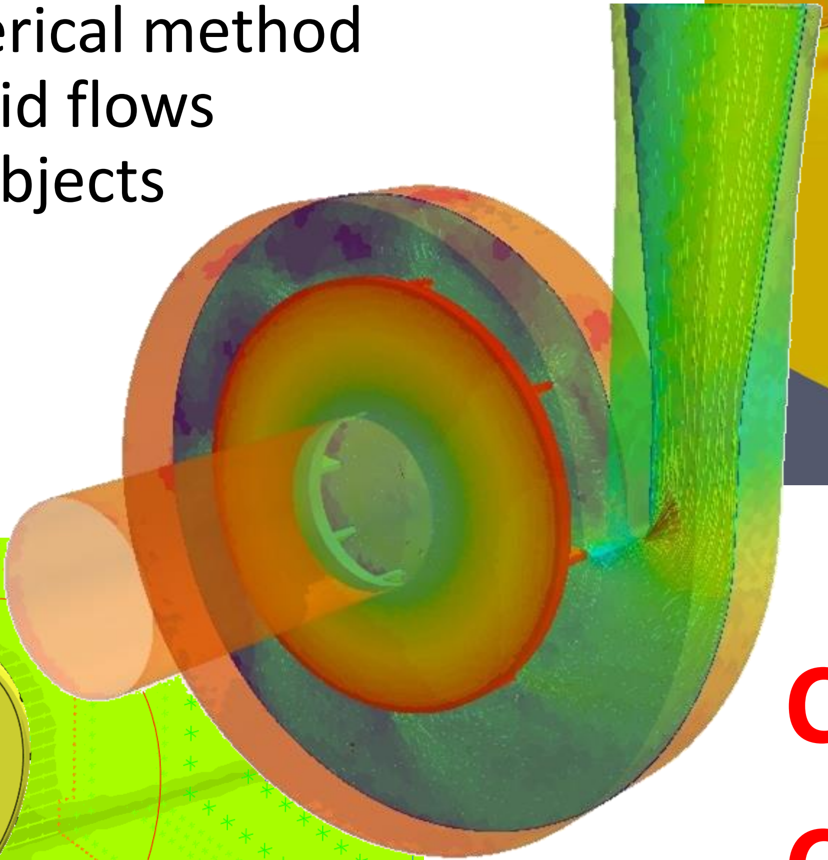
OXFORD BROOKES UNIVERSITY

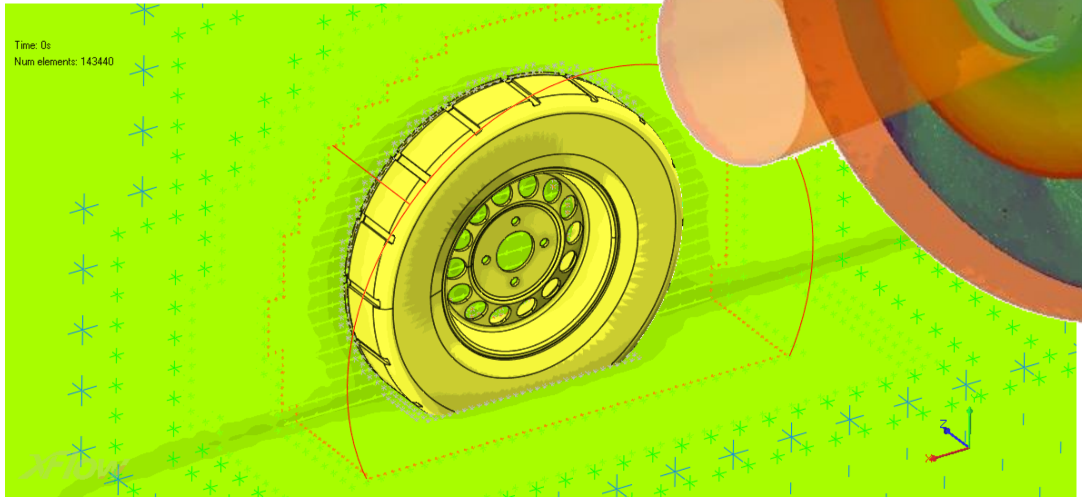
Laurence Marks
07307 588760
laurencemarks64@googlemail.com

What is CFD

- An approximate numerical method used to investigate fluid flows around and through objects







Colours For Directors

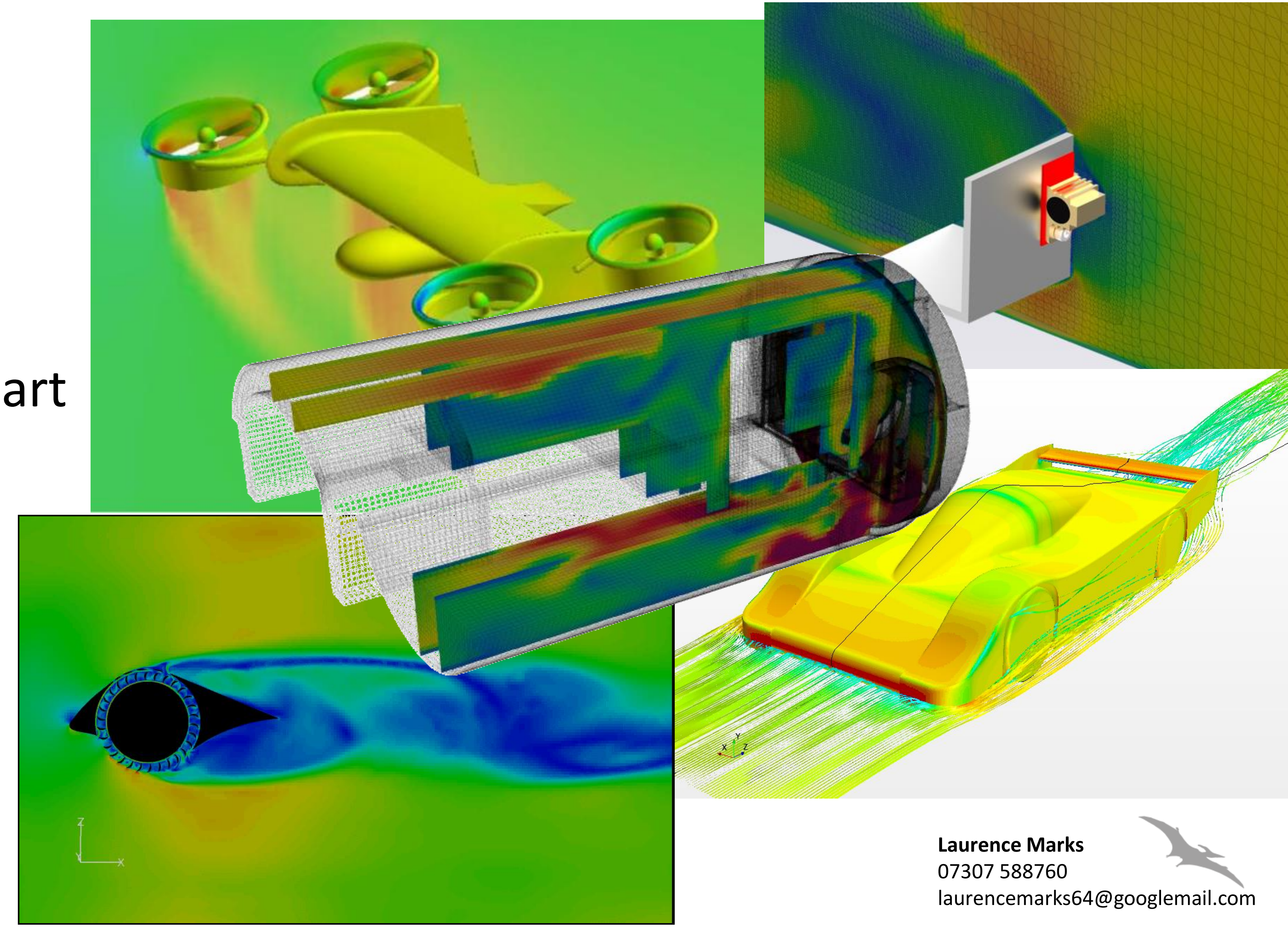
Can’t Find Downforce

OXFORD BROOKES UNIVERSITY

Laurence Marks
07307 588760
laurencemarks64@googlemail.com

What sorts of things are studied using CFD?

- Aerospace external flows
- Automotive external
- Internal flows
- Electronics cooling
- Blood flow through the heart
- Alternative energy
- Pumps
- Buildings

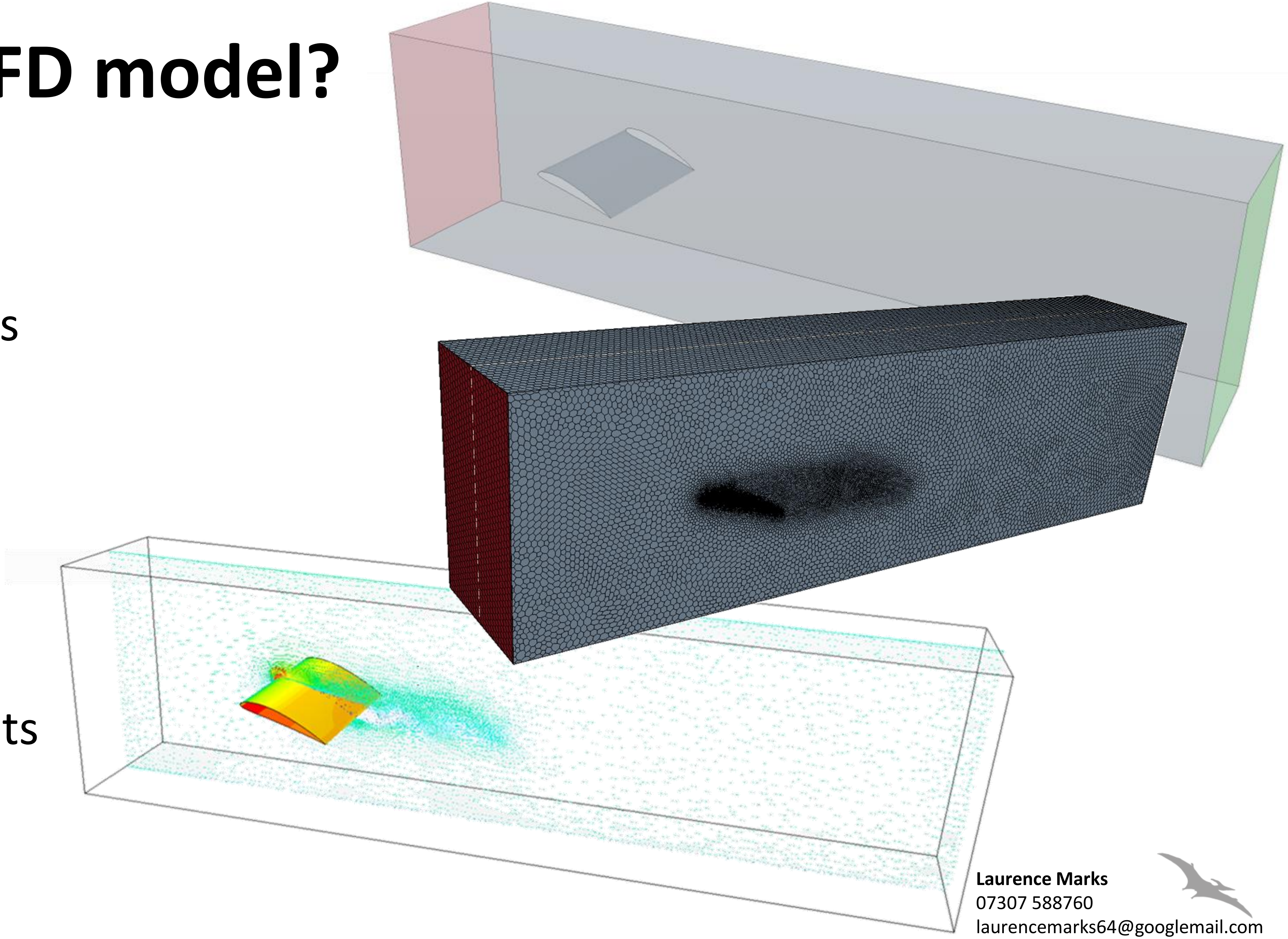


OXFORD BROOKES UNIVERSITY

Laurence Marks
07307 588760
laurencemarks64@googlemail.com

What’s in a CFD model?

- Geometry
- Fluid properties
- Boundary conditions
- Mesh
- Some challenging maths/physics
- Solution control
- And if you are lucky(or good) results

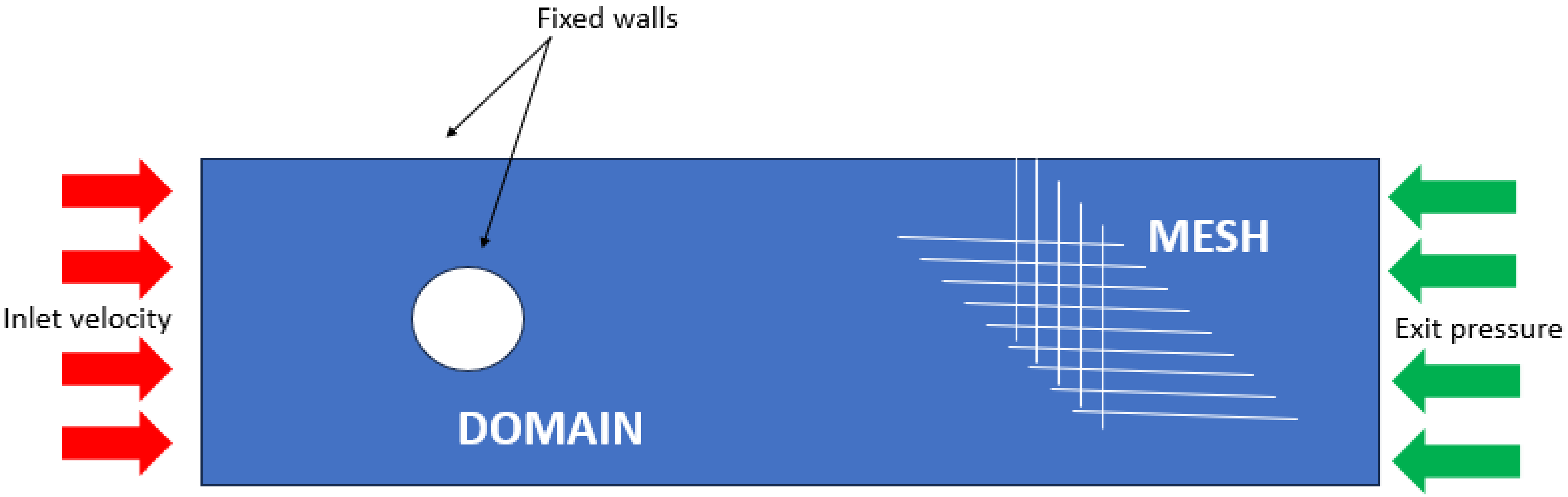


OXFORD BROOKES UNIVERSITY

Laurence Marks
07307 588760
laurencemarks64@googlemail.com

What's in a CFD model?

- Geometry
- Fluid properties
- Boundary conditions
- Mesh
- Some challenging maths/physics
- Solution control
- And if you are lucky(or good) results

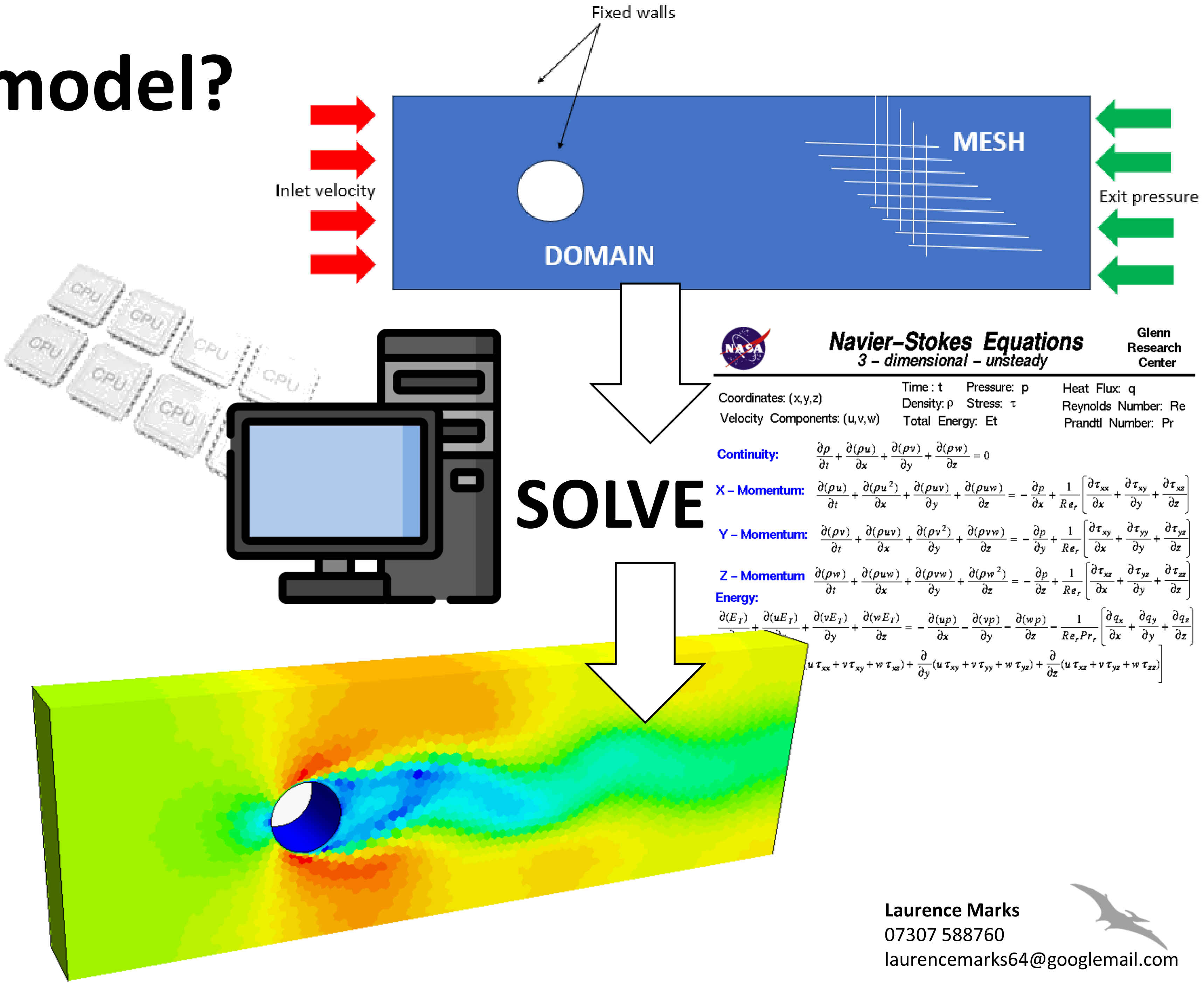


OXFORD BROOKES UNIVERSITY

Laurence Marks
07307 588760
laurencemarks64@googlemail.com

What’s in a CFD model?

- Geometry
- Fluid properties
- Boundary conditions
- Mesh
- Some challenging maths/physics
- Solution control
- And if you are lucky(or good) results

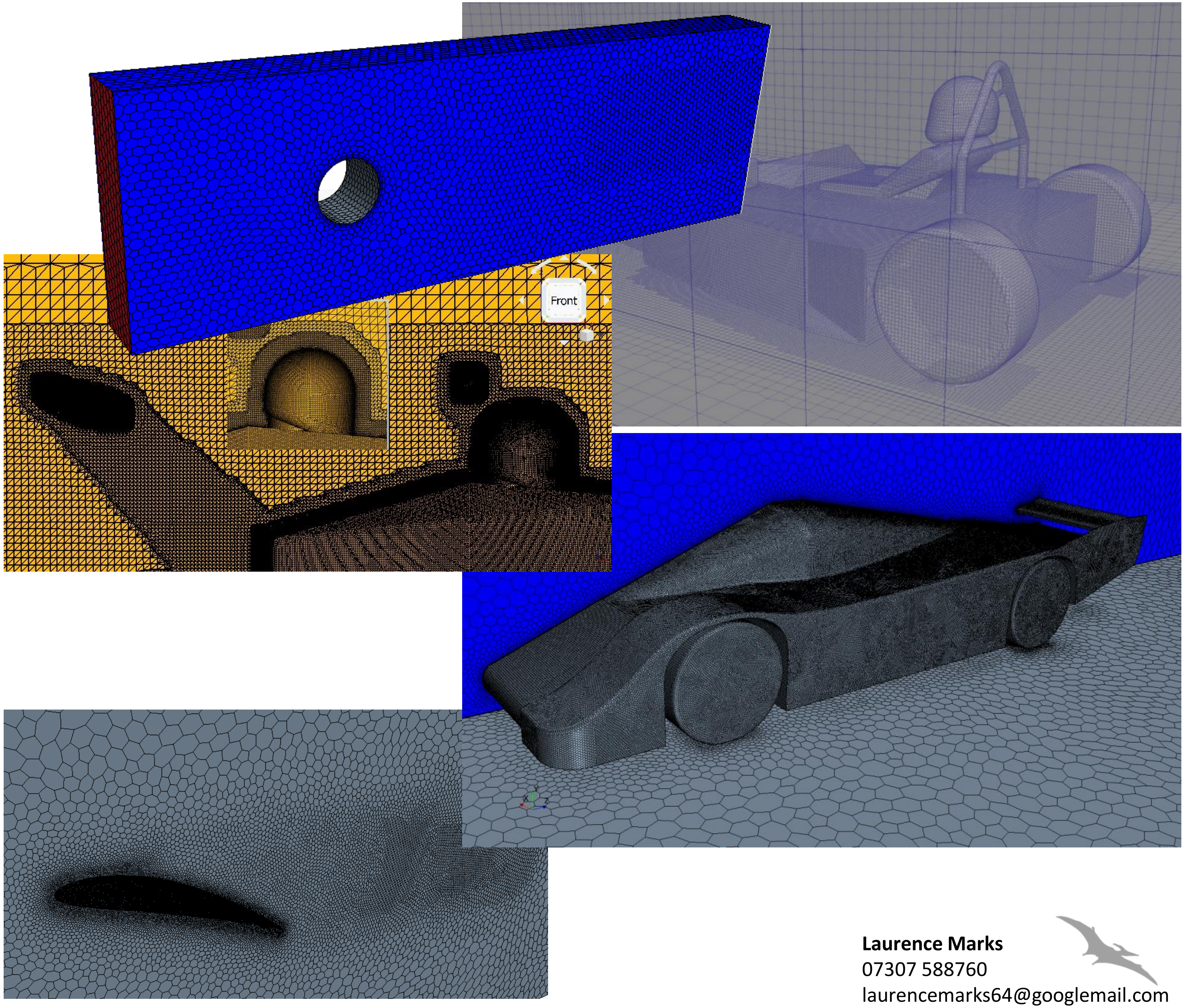


OXFORD BROOKES UNIVERSITY

Laurence Marks
07307 588760
laurencemarks64@googlemail.com

Meshes

- The domain is broken up into cells.
- How you divide the domain up into cells affects resolution, accuracy and speed of convergence.
- It is usual to make finer meshes of cells around objects we are interested in.
- In areas where nothing happens fine meshes waste time and compute power.
- It's a good idea to get the model running with a coarse mesh first



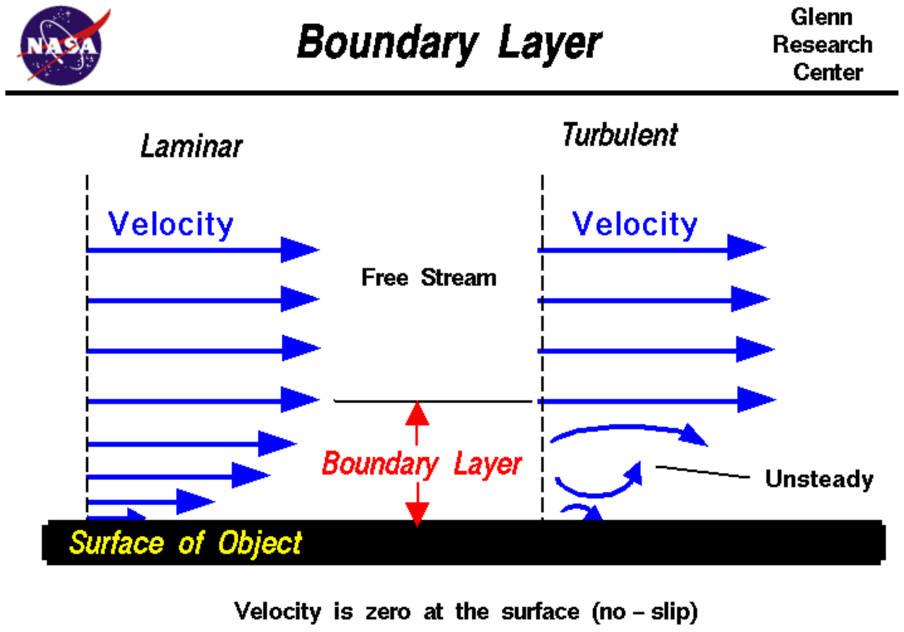
OXFORD BROOKES UNIVERSITY

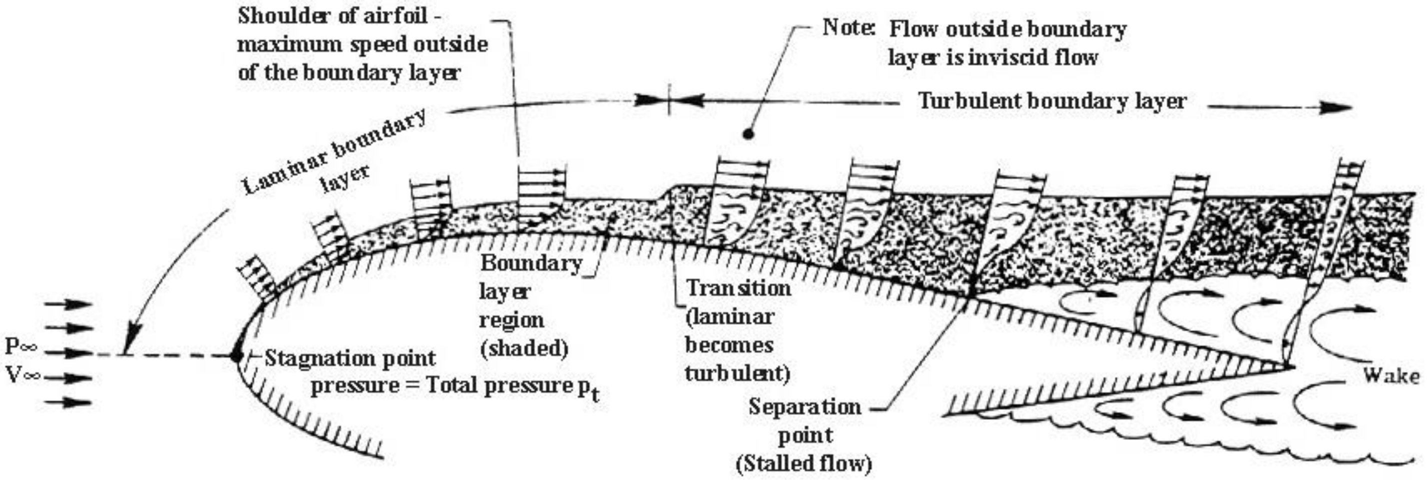
Laurence Marks
07307 588760
laurencemarks64@googlemail.com

Boundary layers

- Are very important

In physics and fluid mechanics, a **boundary layer** is the thin layer of fluid in the immediate vicinity of a bounding surface formed by the fluid flowing along the surface. The fluid's interaction with the wall induces a no-slip boundary condition (zero velocity at the wall). The flow velocity then monotonically increases above the surface until it returns to the bulk flow velocity. The thin layer consisting of fluid whose velocity has not yet returned to the bulk flow velocity is called the velocity boundary layer.





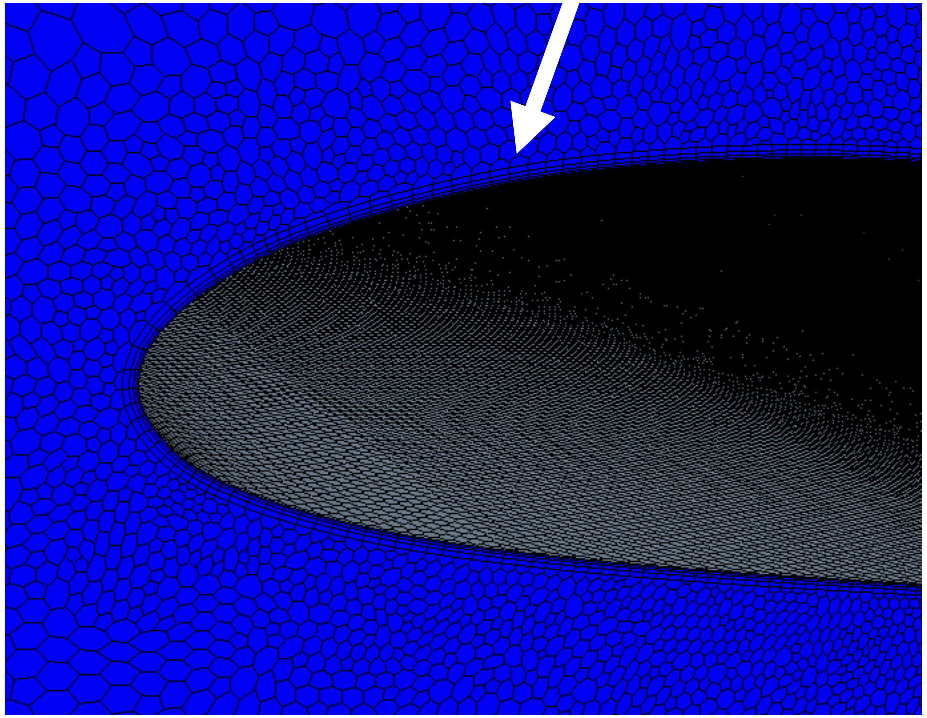
Laurence Marks
07307 588760
laurencemarks64@googlemail.com

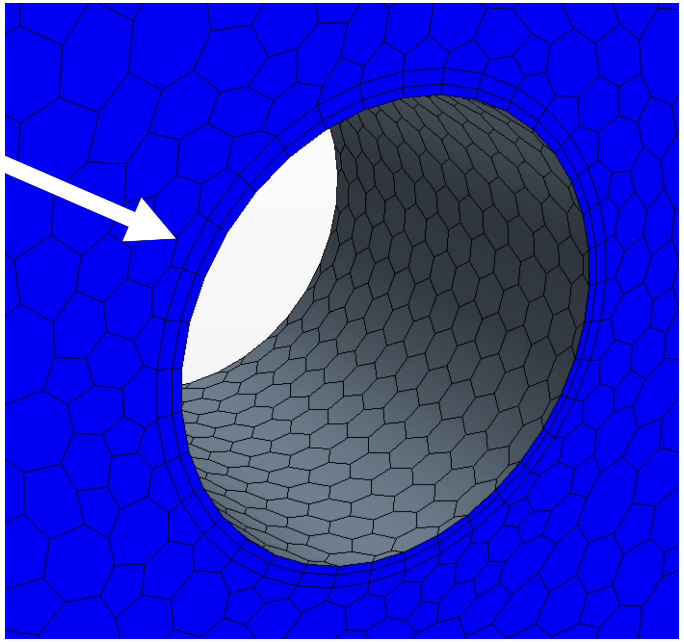
Boundary layers

- So we need to mesh the problem differently to capture the behaviour in these regions

In physics and fluid mechanics, a **boundary layer** is the thin layer of fluid in the immediate vicinity of a bounding surface formed by the fluid flowing along the surface. The fluid's interaction with the wall induces a no-slip boundary condition (zero velocity at the wall). The flow velocity then monotonically increases above the surface until it returns to the bulk flow velocity. The thin layer consisting of fluid whose velocity has not yet returned to the bulk flow velocity is called the velocity boundary layer.

Prism mesher





OXFORD BROOKES UNIVERSITY

Laurence Marks
07307 588760
laurencemarks64@googlemail.com

Turbulence

- Turbulence is a complex phenomena which affects fluid flows.
- It is likely to be important in the studies we are doing.
- We can’t resolve it using arrays of cells we can solve – we use a turbulence model.

> When I meet God, I am going to ask him two questions: Why relativity ? And why turbulence ? I really believe he will have an answer for the first.
>
> Werner Heisenberg

OXFORD BROOKES UNIVERSITY

Turbulence

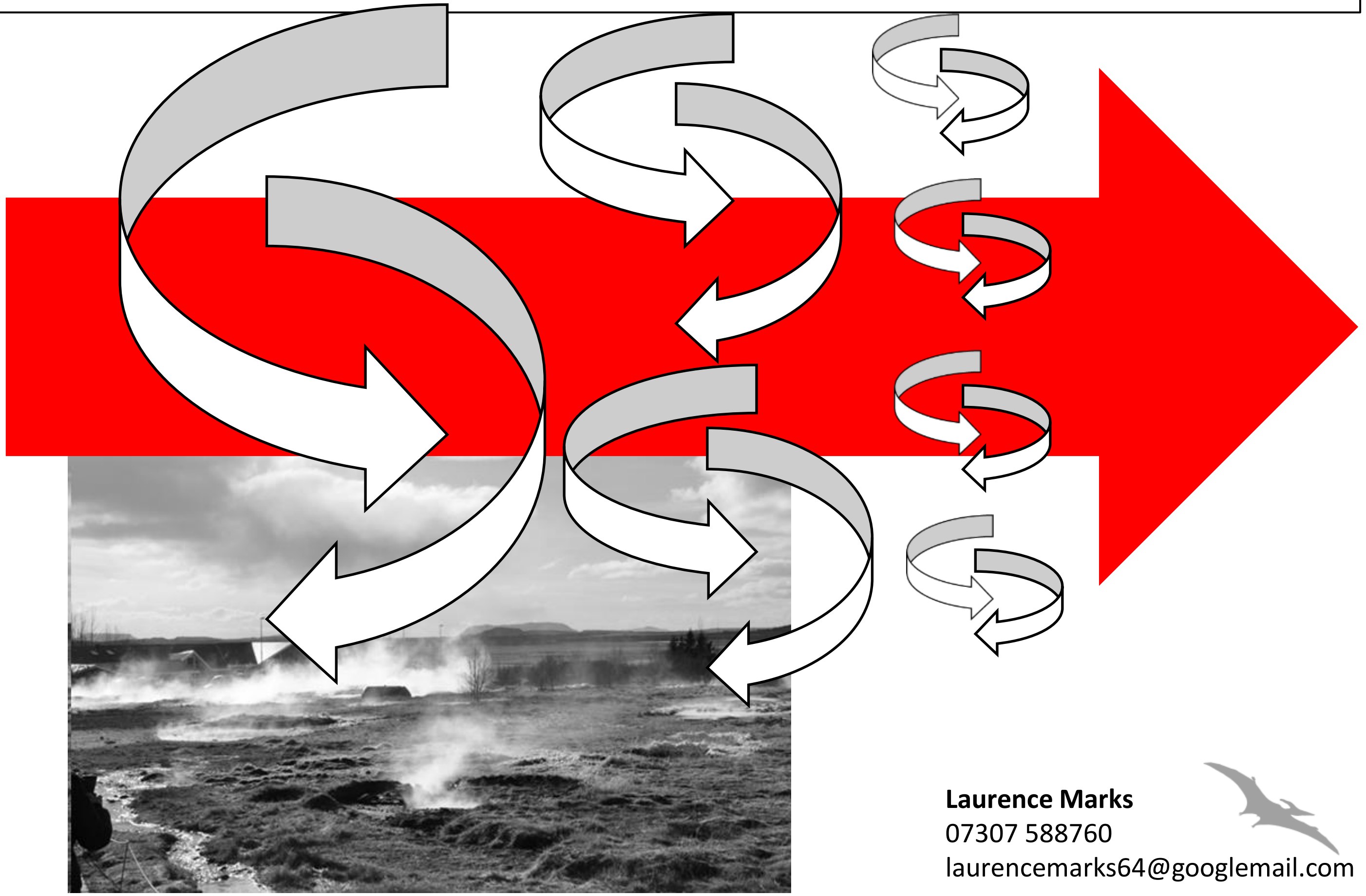
58 languages

Article Talk Read Edit View history Tools

From Wikipedia, the free encyclopedia

For the turbulence felt on an airplane, see Clear-air turbulence. For other uses, see Turbulence (disambiguation).

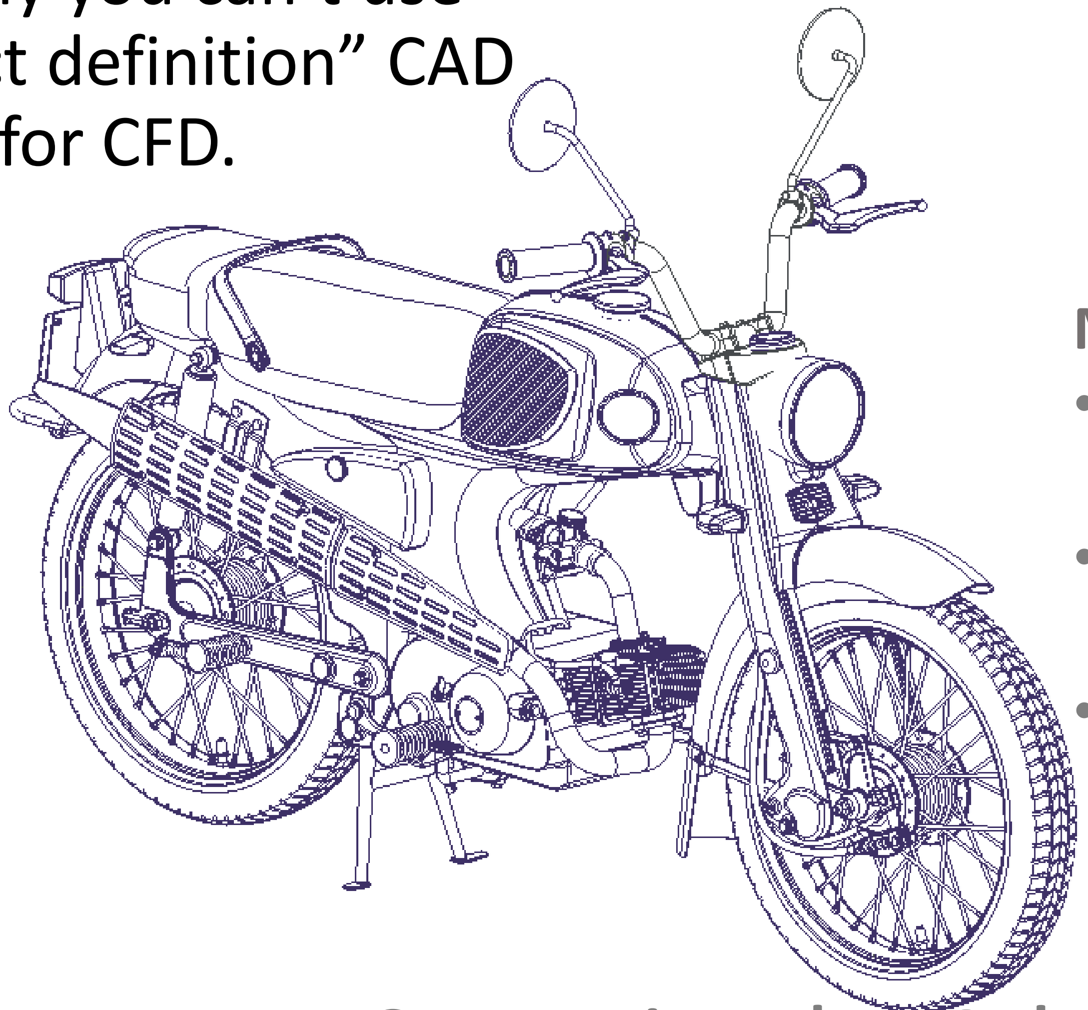
In fluid dynamics, **turbulence** or **turbulent flow** is fluid motion characterized by chaotic changes in pressure and flow velocity. It is in contrast to a laminar flow, which occurs when a fluid flows in parallel layers, with no disruption between those layers.[1]

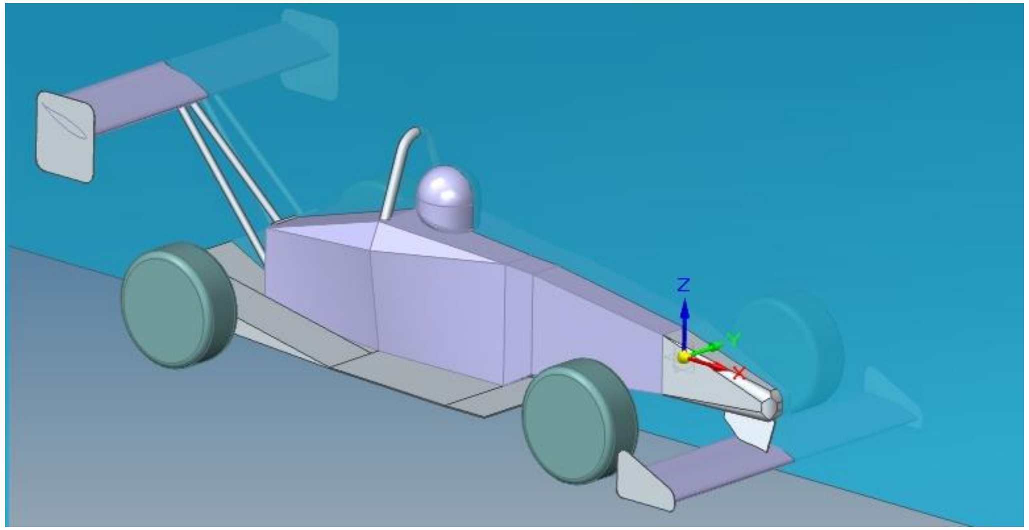


Laurence Marks
07307 588760
laurencemarks64@googlemail.com

Geometry models for CFD.

- Generally you can't use "product definition" CAD models for CFD.





Models need to be simpler

- If you can't resolve the flow round a feature why have it in the model?
- Meshing can be compromised by small parts and badly drawn features
- You always have to balance resolution, accuracy and compute resource and time

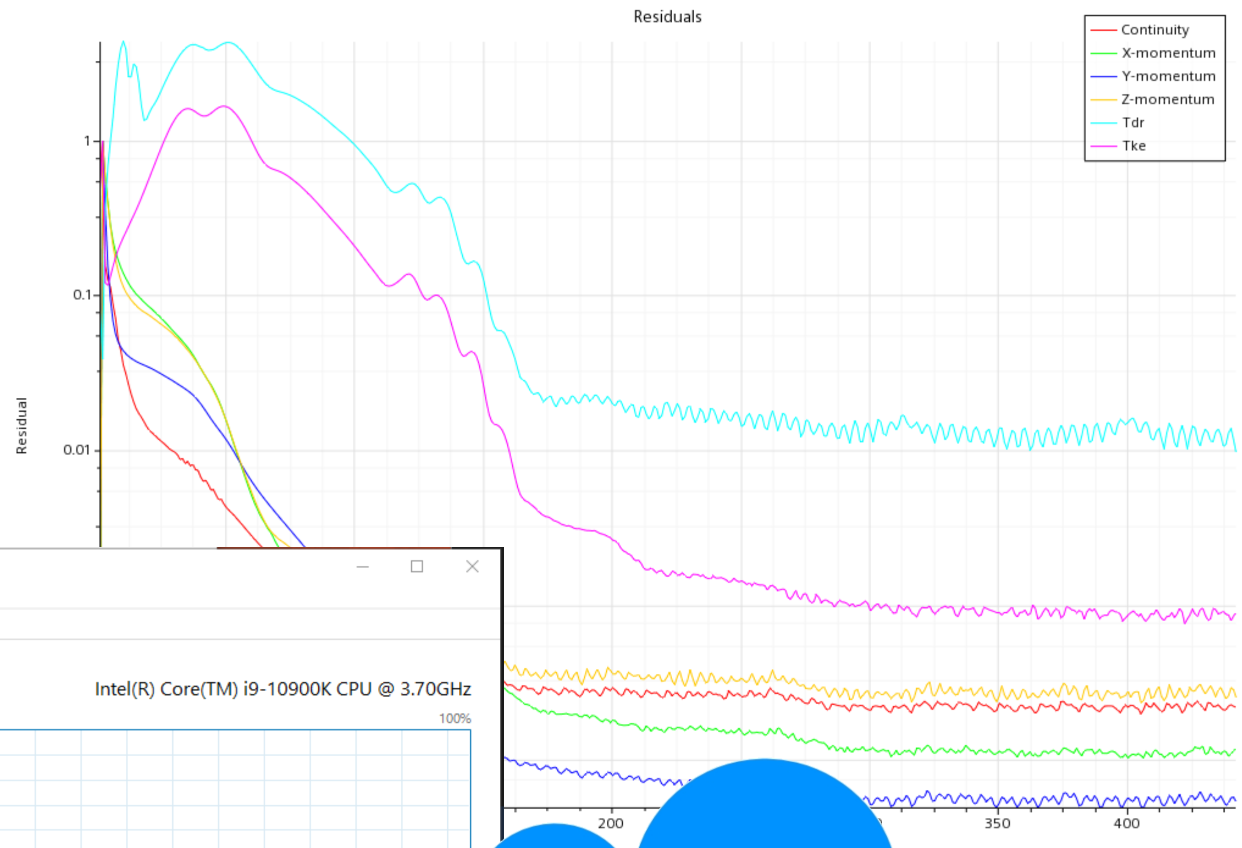
Start simple: Add complexity

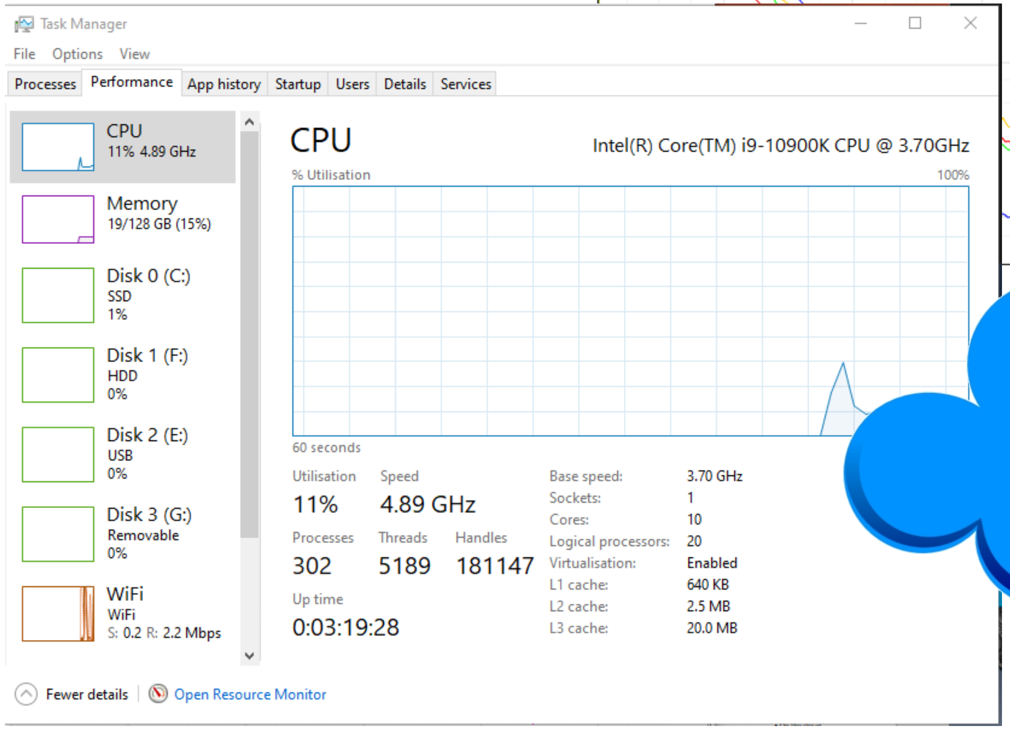
OXFORD BROOKES UNIVERSITY

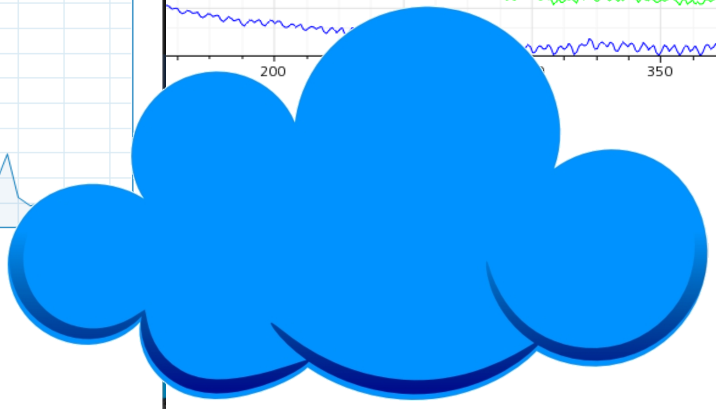
Laurence Marks
07307 588760
laurencemarks64@googlemail.com

Solving

- Equation solving
    - Locally on one or more cores
    - On a cluster
    - On the cloud





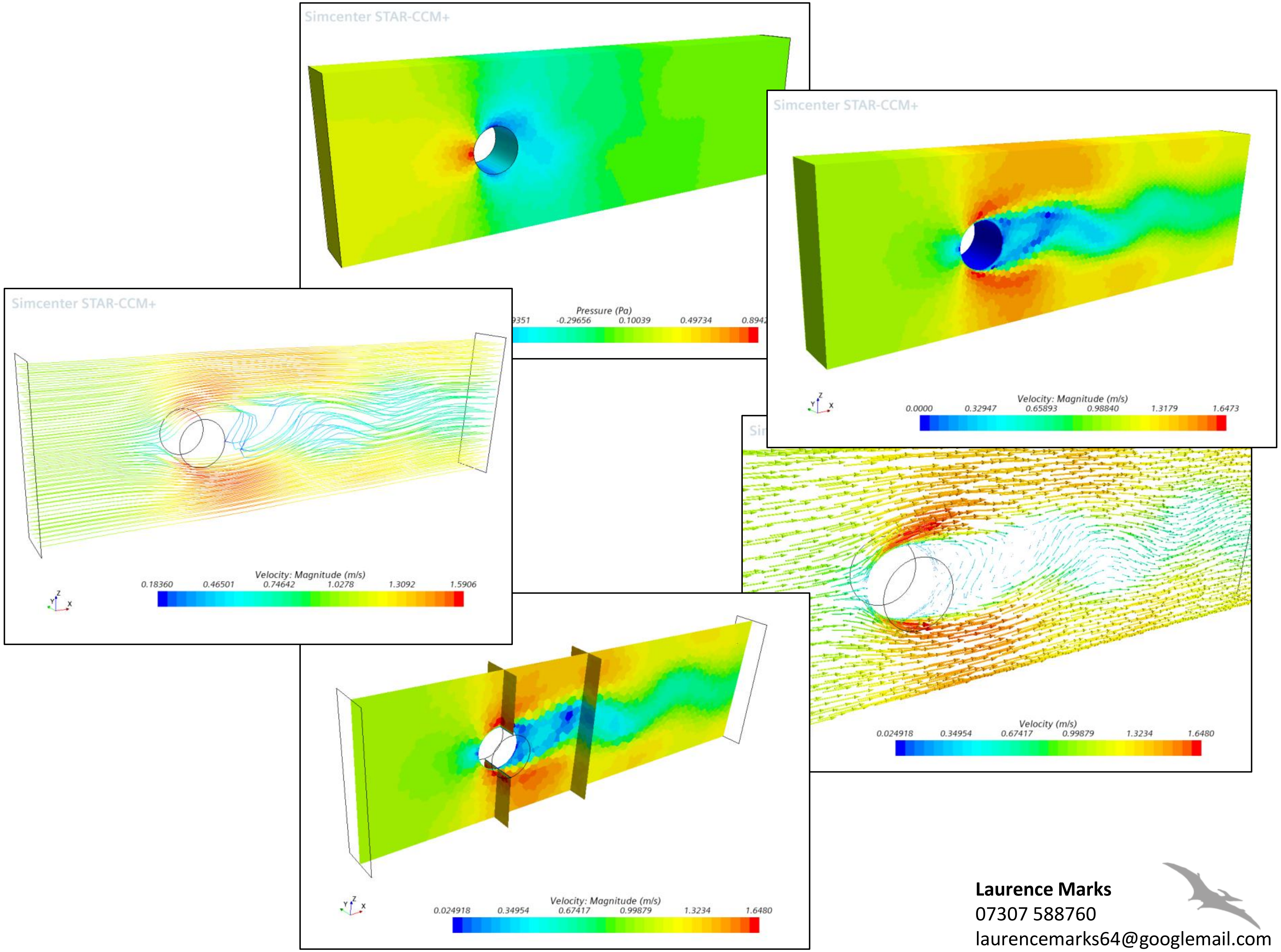


Laurence Marks
07307 588760
laurencemarks64@googlemail.com

Results

- Pressures
  - Forces
- Velocities
- Turbulence values
- Temperatures

- Sections
- Surface plots
- Vector plots
- Streamlines
- Numerical data

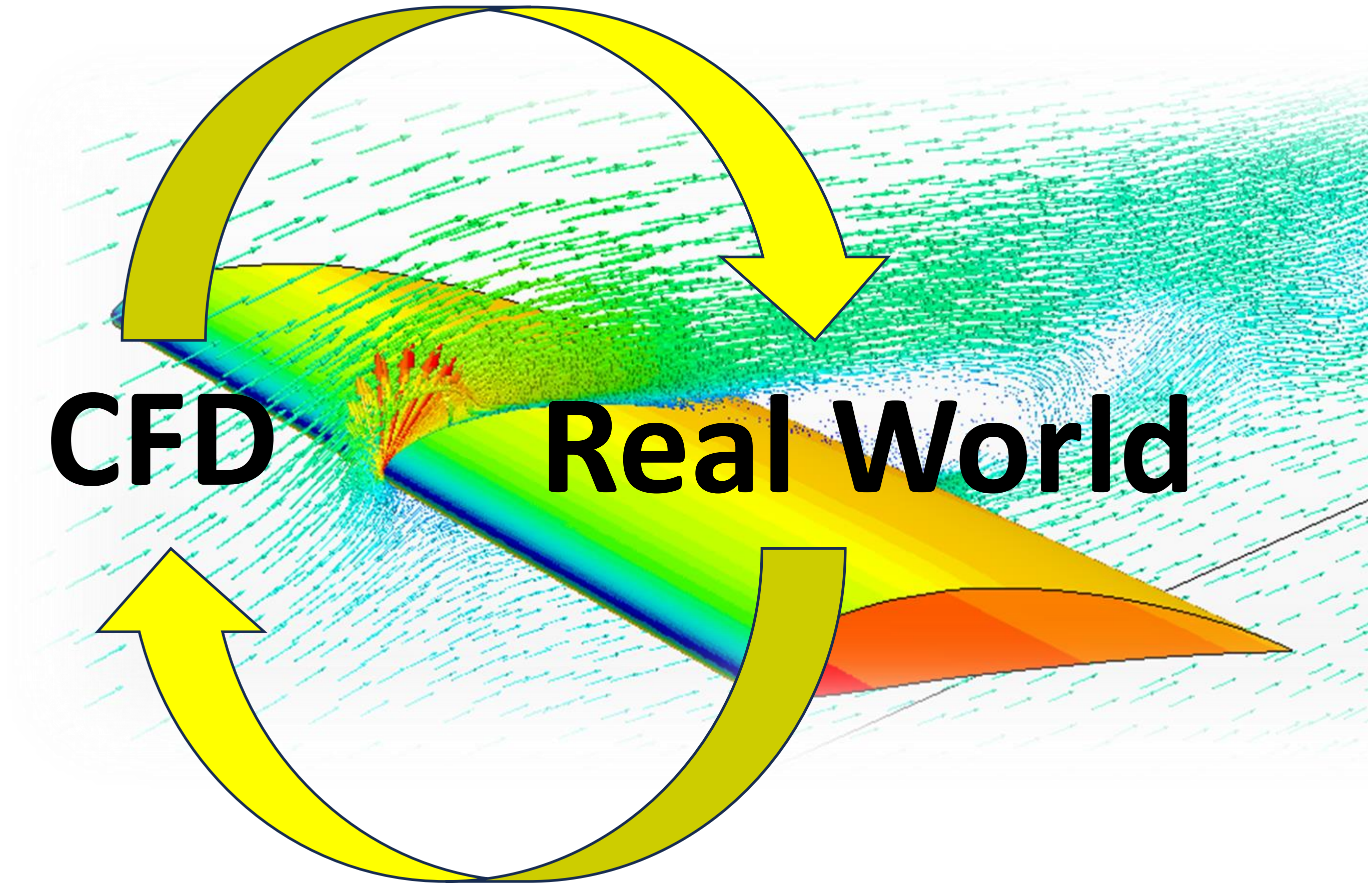


OXFORD BROOKES UNIVERSITY

Laurence Marks
07307 588760
laurencemarks64@googlemail.com

What we do with the results

1. Check them
2. Check them again
3. Compare them with other sources of data – validation
4. Refine/redefine the model
5. Make educated, considered design choices.



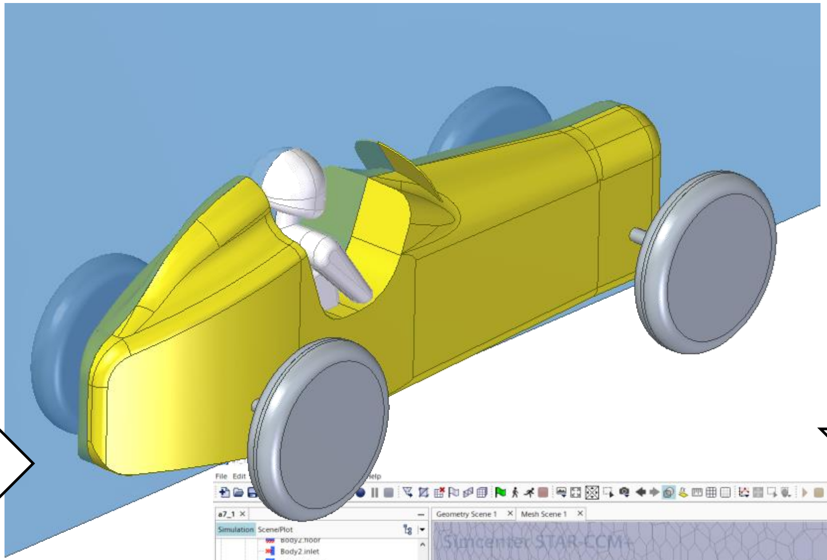
Laurence Marks
07307 588760
laurencemarks64@googlemail.com

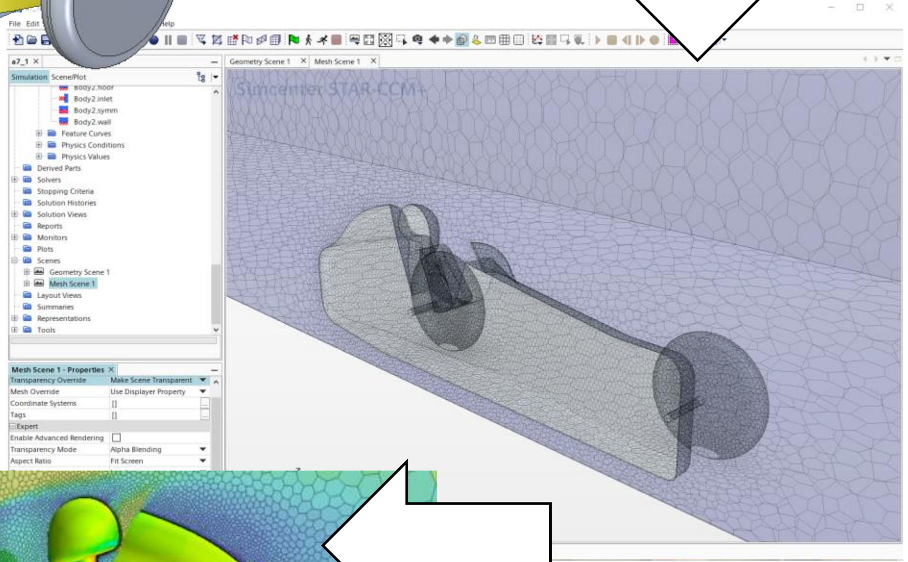
Process – the workflow

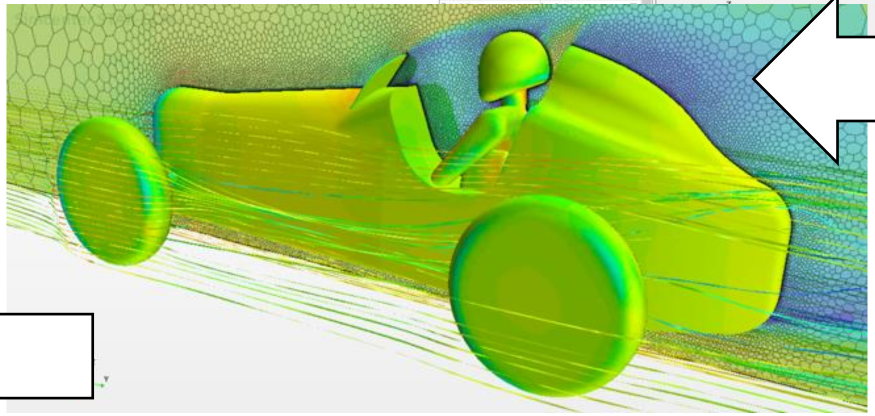
- Defined workflows are good workflows

1. What do I need to find out?
2. How accurate does the answer need to be?
3. How much time have I got?
4. How much compute resource have I got?
5. Honestly, how much do I know about this?
6. Do I know how to interpret what this is telling me?
7. How am I going to validate this stuff?

- Geometry
- Fluids
- Boundary conditions
- System
- Report ??







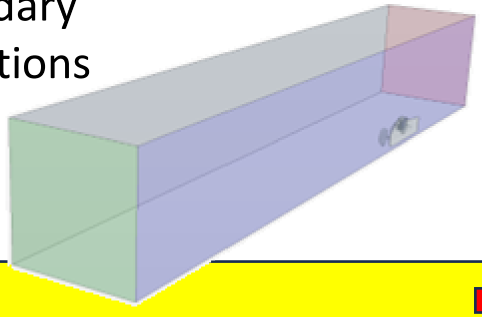
Laurence Marks
07307 588760
laurencemarks64@googlemail.com

Process – the workflow 2

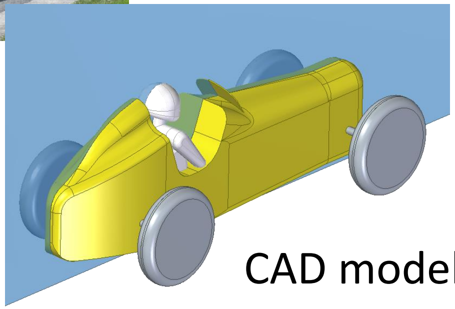
- Defined workflows are good workflows

Problem definition

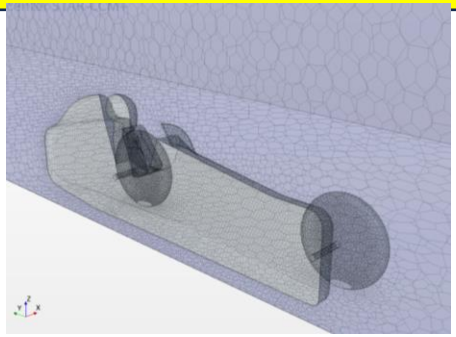
Boundary conditions



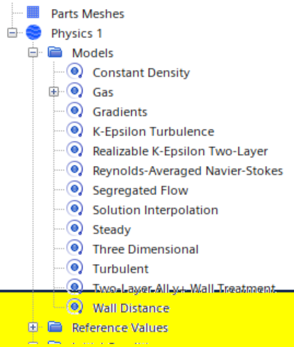
CAD model generation or simplification



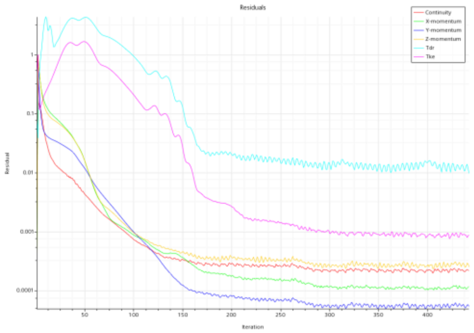
Mesh generation



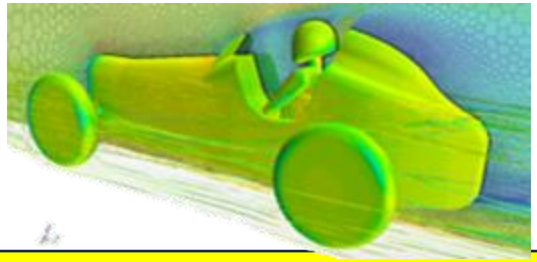
Physics and properties



Solution



Results plotting



Validation

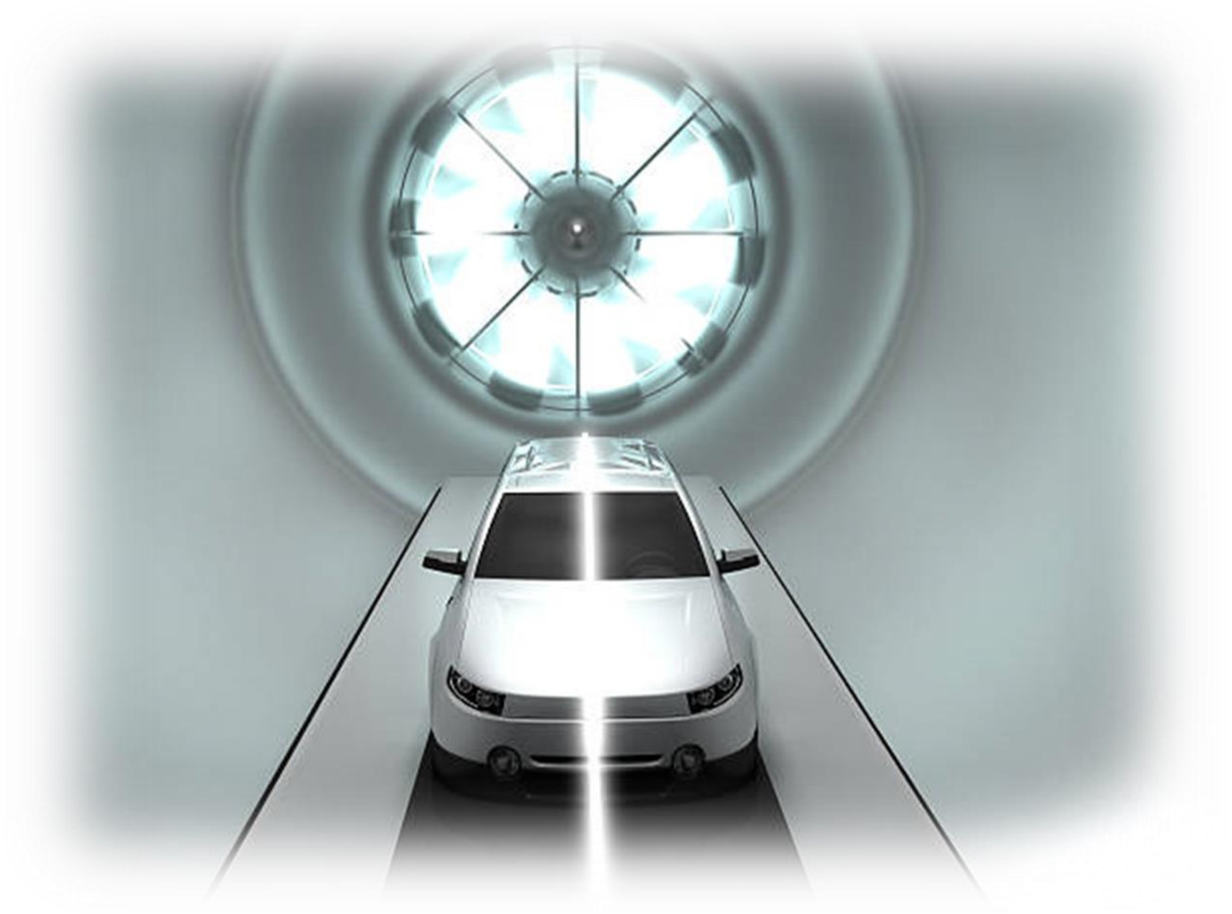
The scope for automation is obvious

Laurence Marks
07307 588760
laurencemarks64@googlemail.com

So lets scrap the wind tunnels..

- We aren’t ready to do that yet…

You’ll understand when you’ve done some CFD.



OXFORD BROOKES UNIVERSITY

Laurence Marks
07307 588760
laurencemarks64@googlemail.com