

# **CFD in the Scope of Race Car Development**

**A tool for Aerodynamic Mastery**

---

Jonas Pangerl

# **Computational Fluid Dynamics**

... is the science of using numerical methods to predict flows based on the governing equations of conservation of mass, momentum, and energy.

# Agenda

## 1 CFD Fundamentals

Discretization, Turbulence  
Modelling, Solving, ...

## 3 A Workflow For Formula Student

All Steps from CAD to  
Postprocessing

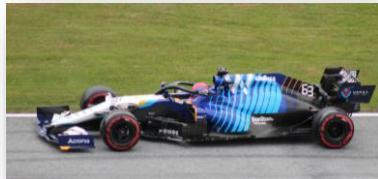
## 2 Race Car CFD

Key Aspects of Modelling Race  
Car Components

## 4 Validation and Correlation

Accuracy, Understanding and  
Optimizing the CFD Setup

# About Me



---

**Rennstall  
Esslingen**

Team Leader Aerodynamics

**Williams  
Racing**

Aerodynamicist,  
Master Thesis about High  
Fidelity CFD

**Sauber  
Motorsport**

CFD Correlation Engineer

# CFD For Race Car Development

## Aerodynamic Optimization

Optimize Aerodynamic Properties, such as Downforce and Drag

## Cost and Time Efficient

No physical Models and very quick turn-around time

## Detailed Flow Analysis

Insights into Flow Field, Pressure Distribution and Turbulence Properties

---

## Real-World Conditions

Modeling of real-world Conditions such as curved Flow, Ground Contact and Bouncing

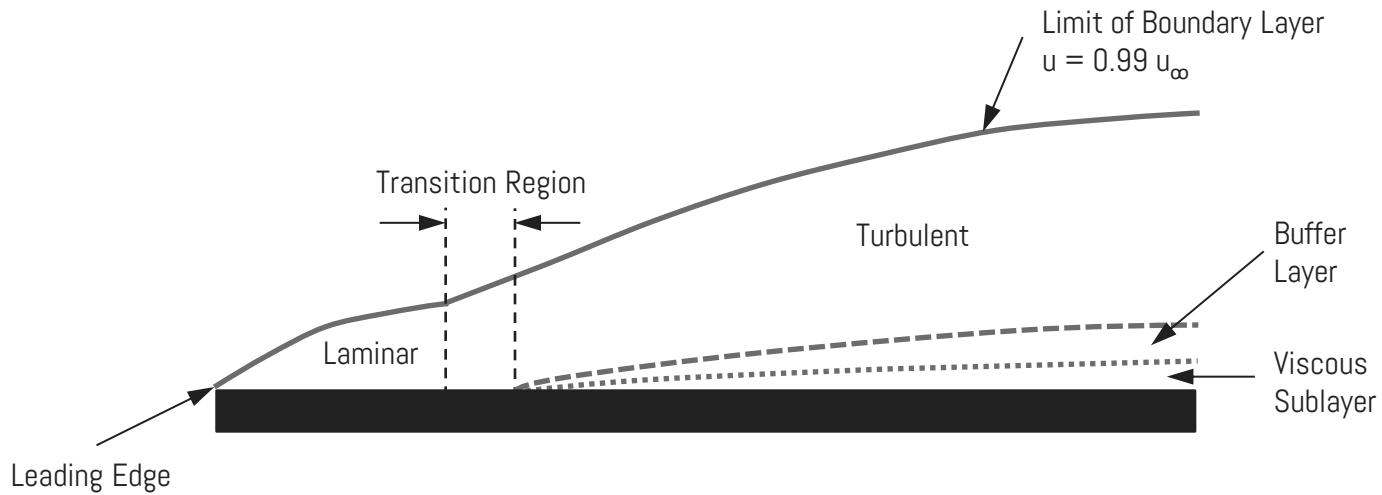
## Radical Design Exploration

Evaluation of big conceptual Changes and radical Concepts

## Adjoint and AI Enhancements

Sensitivity Analysis, Automated Design Process, AI Analysis and Predictions

# Excurs: Boundary Layer Theory



1

CFD

Fundamentals

---

# Discretization

---

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \underline{u}) = 0$$

$$\frac{\partial}{\partial t}(\rho H) + \nabla \cdot (\rho \underline{u} H) = -\nabla \cdot \underline{g} + \frac{Dp}{Dt} + \underline{\tau} : \nabla \underline{u}$$

$$\frac{\partial}{\partial t}(\rho \underline{u}) + \nabla \cdot (\rho \underline{u} \otimes \underline{u}) = -\nabla p + \nabla \cdot \underline{\tau} + \rho \underline{g}$$

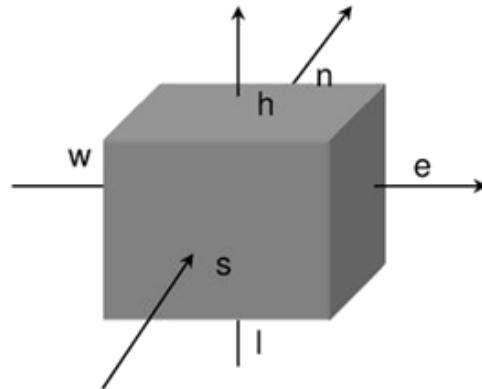
- » Numerical Techniques such as **Finite Volume Method** (FVM) are used to compute the quantities in each control volume
- » The continuous space is described by a finite set of equations that can be handled by the computer
- » The finer the discretization, the more accurate the solution

The state and changes of a fluid or gas can be described mathematically through the **Navier-Stokes Equations**

- » These are partial differential Equations (PDEs)
- » PDEs can not be solved analytically for real-world problems

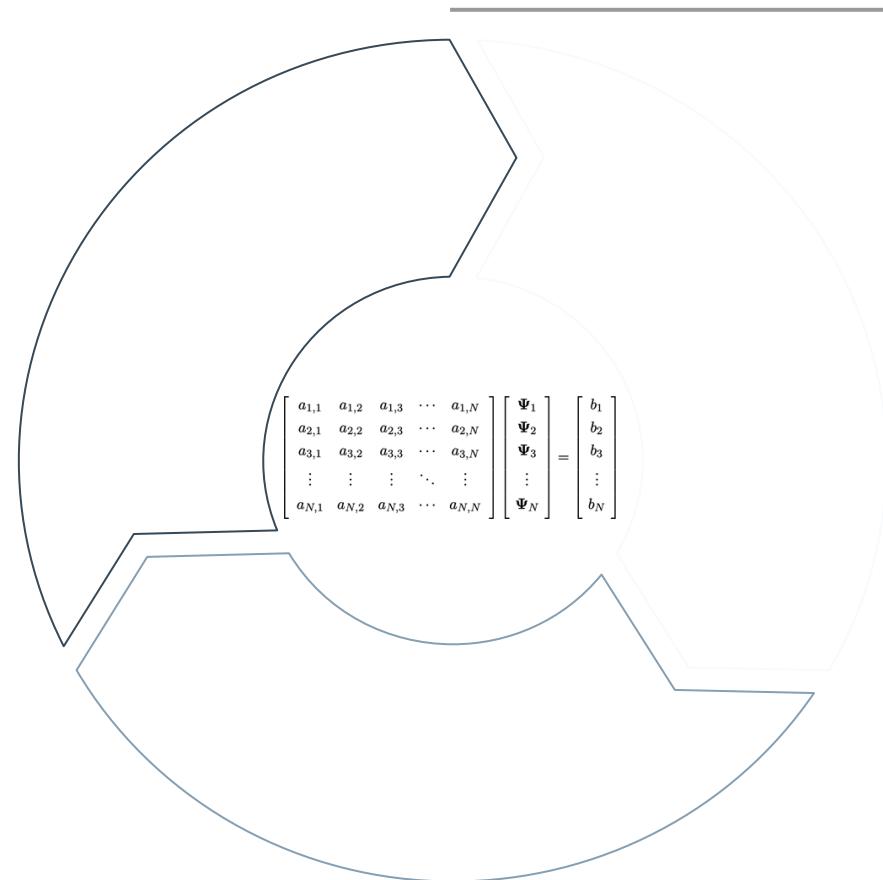
Solution:

Approximate the continuous equations through discretization with a finite set of elements



# Solving Process

- Discretization turned differential equations into a set of algebraic equations
  - Momentum Equation links velocity components from one cell to the neighbouring cells
  - Continuity Equation links pressure and velocity
  - For each cell, we must compute  $u_i, u_j, u_k, p, \dots$
- » pressure-velocity link and nonlinear terms do not allow to solve the system in one step
- » Iterative Solving Process


$$\begin{bmatrix} a_{1,1} & a_{1,2} & a_{1,3} & \cdots & a_{1,N} \\ a_{2,1} & a_{2,2} & a_{2,3} & \cdots & a_{2,N} \\ a_{3,1} & a_{3,2} & a_{3,3} & \cdots & a_{3,N} \\ \vdots & \vdots & \vdots & \ddots & \vdots \\ a_{N,1} & a_{N,2} & a_{N,3} & \cdots & a_{N,N} \end{bmatrix} \begin{bmatrix} \Psi_1 \\ \Psi_2 \\ \Psi_3 \\ \vdots \\ \Psi_N \end{bmatrix} = \begin{bmatrix} b_1 \\ b_2 \\ b_3 \\ \vdots \\ b_N \end{bmatrix}$$

# Solving Process

---

## Segregated Solver

---

Variables solved sequentially.  
E.g. First u, then p

---

No direct link between variables in equations  
Needs correction steps to fulfill continuity equation

---

Memory efficient, less computational power per iteration (time  
to solve equations in one iteration)

---

May converge slower (more iterations)

---

## Coupled Solver

---

Variables solved simultaneously

---

Accounts for interaction between variables

---

Requires more memory and more computational power per  
iteration

---

Converges faster (fewer iterations)

---

# Solving Process

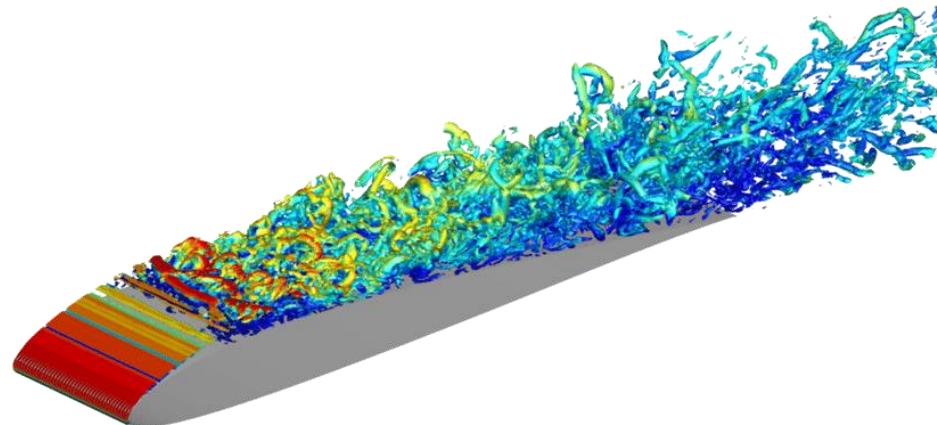
---

Possible to solve Navier-Stokes equations directly, but it requires:

- Resolving all turbulent structures in the flow
  - Very fine meshes
  - Time-resolved simulations
- » Not possible for most engineering applications in feasible time and with usual computational resources

What can be done to overcome that issue:

- Model (some) turbulent structures → coarser meshes
- Calculate time-averaged result



# Solving Process

## Transient

Time-resolved  
Resolved time-dependant flow structures

High computational cost

Turbulence resolved or minimally modelled

Best for dynamic flows

Needs a very fine mesh

## RANS

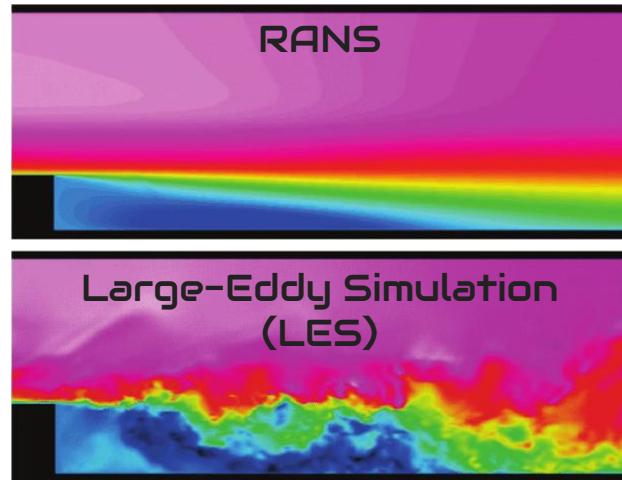
Time-averaged  
Mean flow characteristics, does not capture time dependant effects

Low computational cost

Modelled Turbulence

Best for steady-state flows

Coarser meshes possible



# Turbulence Modelling

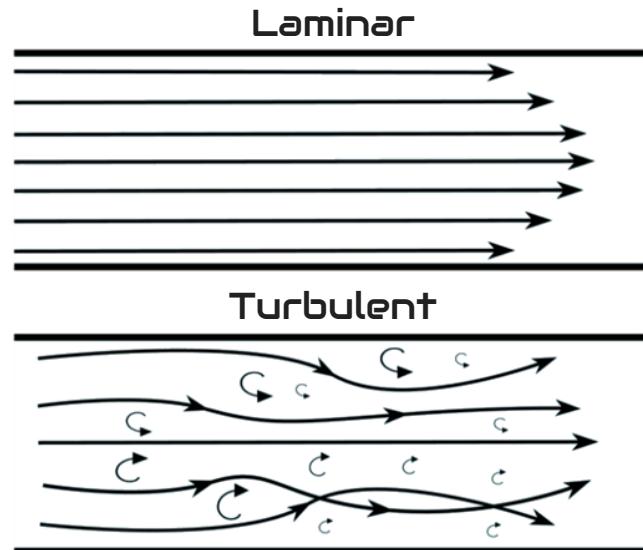
---

How can the effect of turbulence on the flow be modelled?

- Boussinesq Approximation: Turbulent stress can be expressed like shear stress
- » Dynamic viscosity in the equations is replaced by the sum of laminar viscosity and turbulent viscosity

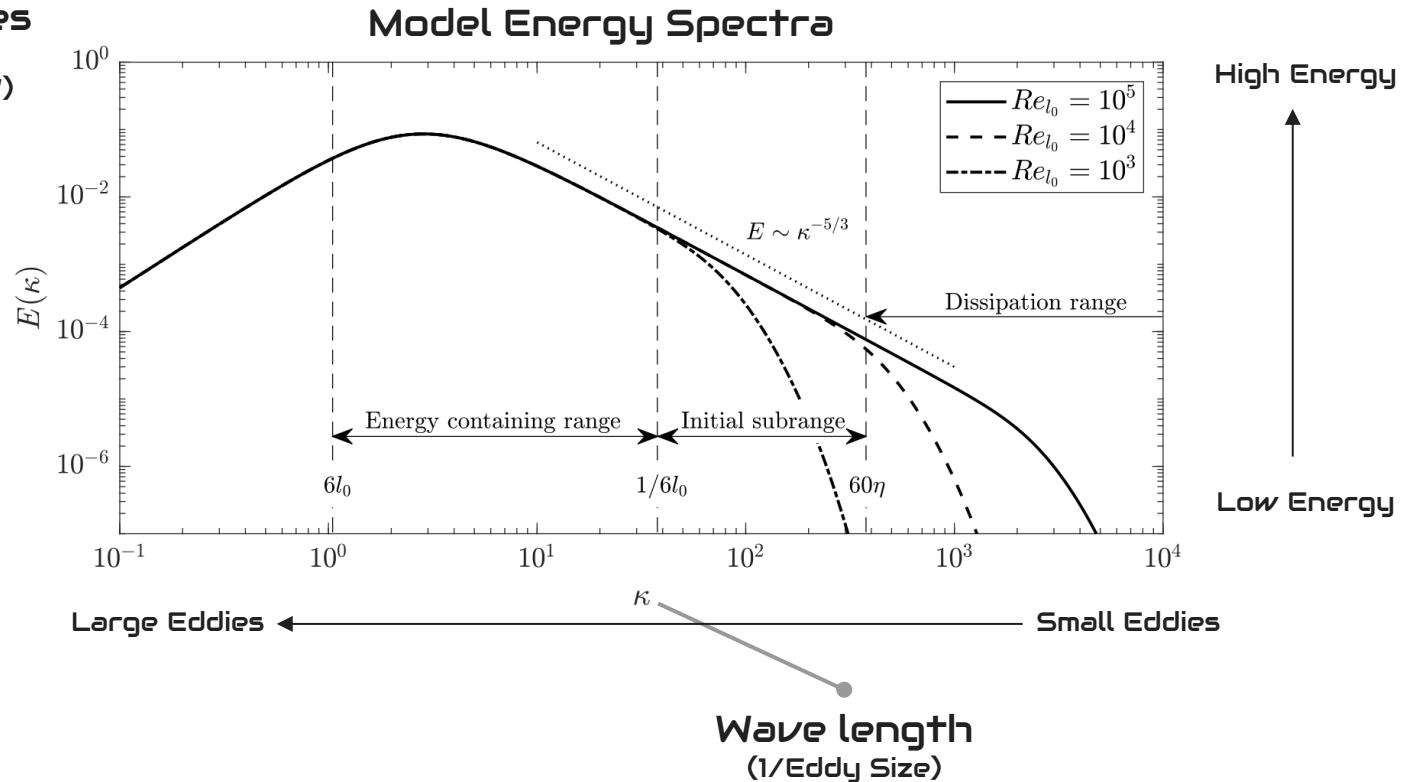
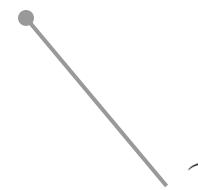
How can the turbulent viscosity be calculated?

- Turbulence models are used to estimate the amount of turbulence with information from the flow quantities (velocity, shear)
- Those models try to mimic the effect of turbulence on the flow quantities and are based on experiments which are used to "tune" the coefficients used in the model

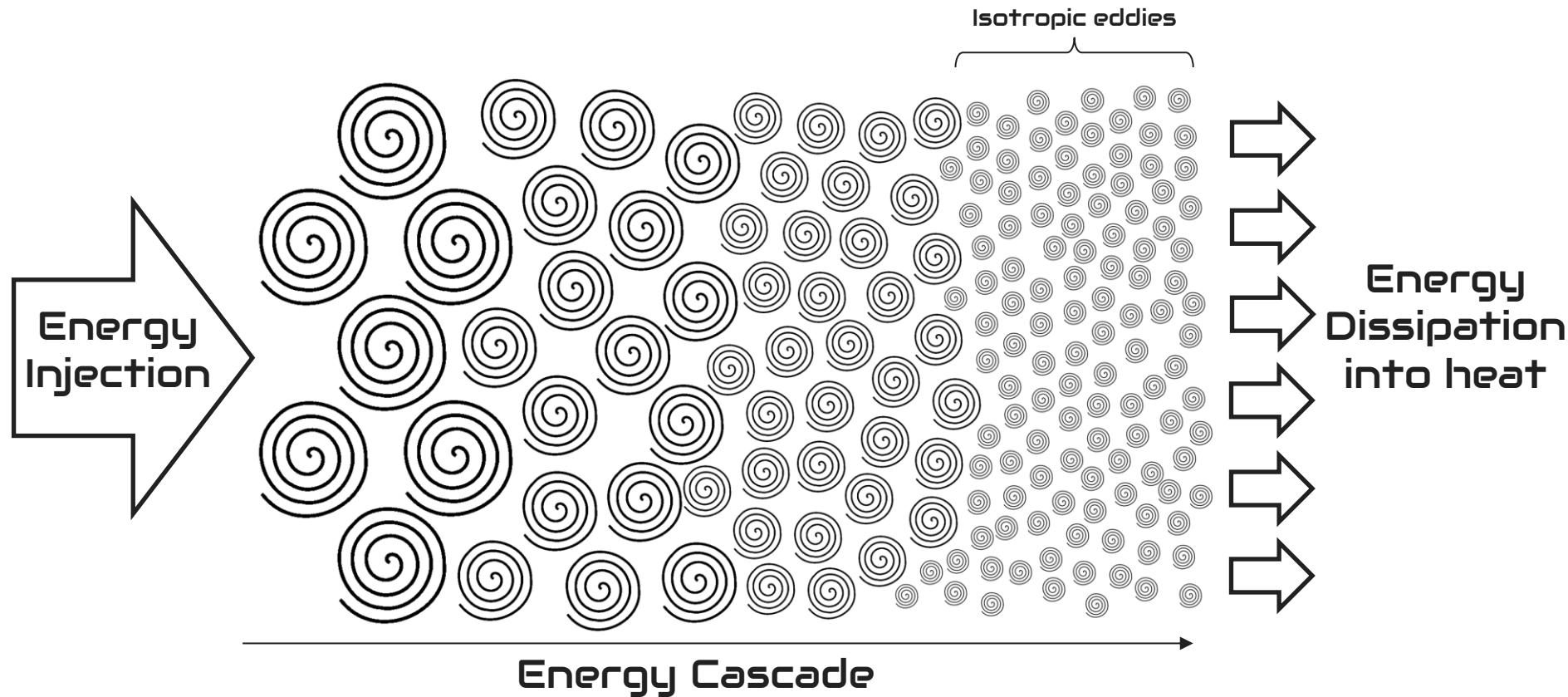


# Turbulence Modelling

**Energy of Eddies**  
(Eddy = „single turbulent structure“)



# Turbulence Modelling

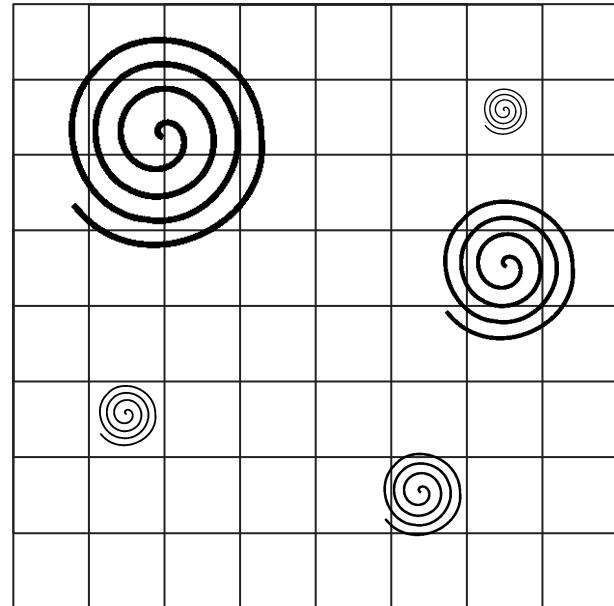


# Turbulence Modelling

---

## Large-Eddy Simulation

- Transient Simulation
  - Resolve large, anisotropic vortices that carry the most energy
  - Model small, isotropic vortices that dissipate energy
  - Turbulence Model "decides" what can be resolved and what is modelled based on mesh size
- » LES turbulence models are called "Sub-Grid Scale" (SGS) Models
- » These models are only designed to deal with very small, mostly energy dissipating, vortices
- » Resolving turbulence inside the boundary layer of the wall needs extremely fine meshes (up to 99% of all cells are placed inside the boundary layer)



# Turbulence Modelling

## RANS Turbulence Models

Different models are available to estimate the turbulent viscosity. Focus here is on Two-Equation Models

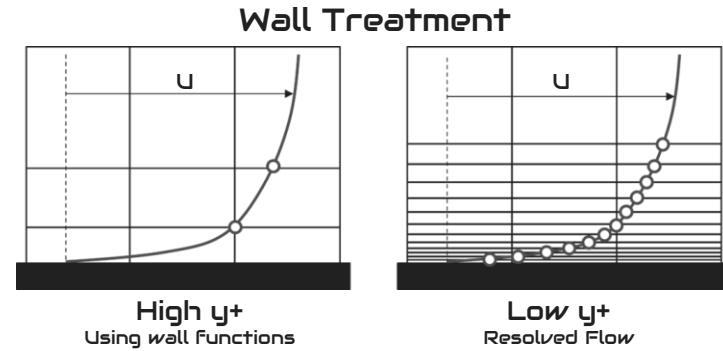
$$k\text{-epsilon Model: } \mu_T = \rho C_\mu \frac{k^2}{\epsilon}$$

$$k\text{-omega Model: } \mu_T = \rho \frac{k^2}{\omega}$$

k: turbulent kinetic energy

$\epsilon$ : turbulent dissipation rate

$\omega$ : specific rate of dissipation



- » Two additional transport equations needed to estimate the turbulent viscosity
- » More accurate than One-Equation Model(s) in complex flow and less expensive than other approaches (E.g. Reynolds-Stress Model with 7 Equations)
- » Resolving Boundary Layers needs only a small spacing of the mesh in wall-normal direction
- » The boundary layer can also be fully modelled by wall-functions which needs much less cells

# Turbulence Modelling

---

	Realizable k-epsilon	SST Menter k-omega
<b>Formulation</b>	Improved and more physically accurate than standard k-epsilon	Blends between k-omega near walls and k-epsilon in far field
<b>Wall Treatment</b>	Needs additional models for low $y+$ *	All $y+$
<b>Transition</b>	No**	No**
<b>Adverse Pressure Gradients</b>	Moderate	Excellent
<b>Free-shear Flows</b>	Good	Moderate
<b>Separated Flows</b>	Poor	Excellent
<b>Application</b>	Not recommended for race car CFD. Lag elliptic-blending k-epsilon might be better, but computationally more expensive	Recommended for race car CFD

\*Already implemented in most commercial CFD codes,   \*\*Possible with additional transition model

# Turbulence Modelling

---

## Advantages

### Steady-State (RANS)

- » Low computational cost
- » Predicts mean flow well
- » Low mesh requirements

## Disadvantages/Problems

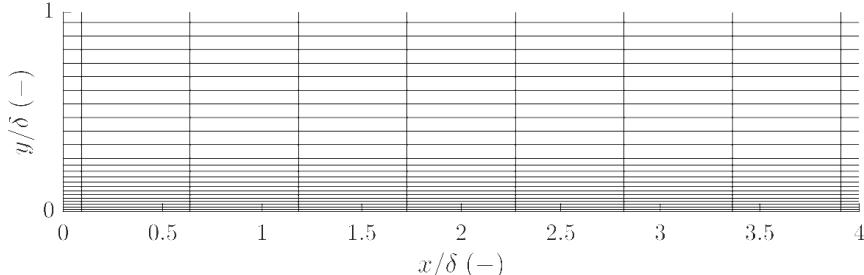
- » Low accuracy for transient phenomena
- » Limited in separated flows/high streamline curvature
- » Empirical dependency

### Transient (LES)

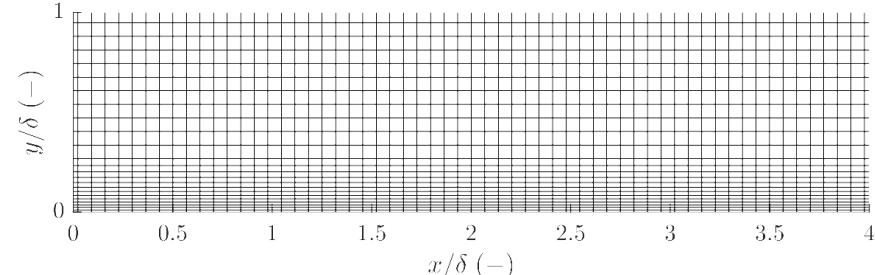
- » Captures detailed turbulent structures
- » Great for unsteady and complex flows

- » High computational cost
- » Difficult in handling near wall flow
- » **High mesh requirements**

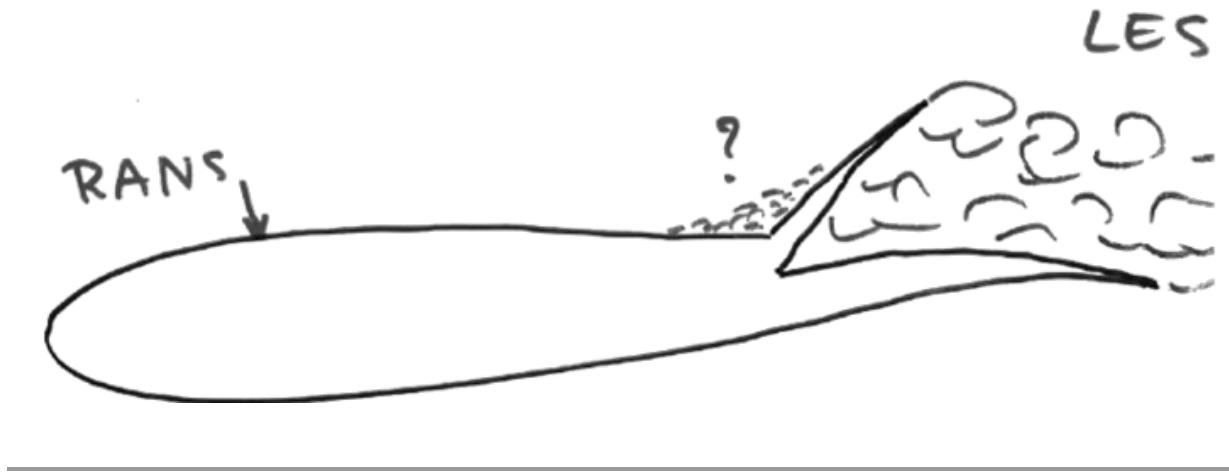
RANS near-wall Mesh



LES near-wall Mesh



# Turbulence Modelling



---

Original concept sketch of C.P. Spalart for his **Detached Eddy** approach (1997)

# Turbulence Modelling

---

Hybrid RANS-LES Approach: **Detached Eddy Simulation (DES)**

- Model near-wall flow with RANS model
- RANS uses Spalart-Allmaras or k-omega Turbulence Model
- Separated off-body flow with LES
- » Removes the requirement for very fine meshes inside the boundary layers
- » Overcomes weaknesses of RANS in the off-body flow
- » Boundary Layers are modelled with RANS limitations
- » Problems with (the position of) the interface between RANS and LES regions

**Delayed-Detached Eddy Simulation (DDES)**

- Ensures the boundary layer is always treated by RANS and the interface to the LES region lies outside of the boundary layer
- Should always be used instead of the original DES
- » Boundary Layers are modelled with RANS limitations

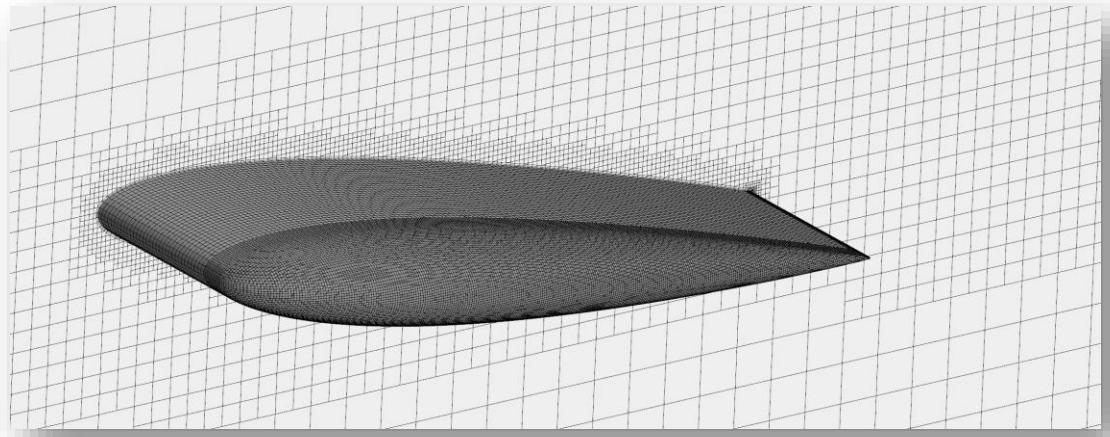
**Improved-Delayed-Detached Eddy Simulations (IDDES)**

- Also known as Wall-Modelled LES (WMLES)
- Places the RANS-LES interface just above the viscous sublayer of the boundary layer flow
- Non-turbulent part of near-wall flow is treated by the RANS model
- Operates in IDDES mode if mesh is fine enough near walls, otherwise in DDES mode
- » Highest accuracy of hybrid RANS-LES models
- » Higher mesh requirements than DDES and difficult to control the interface position

# Meshing

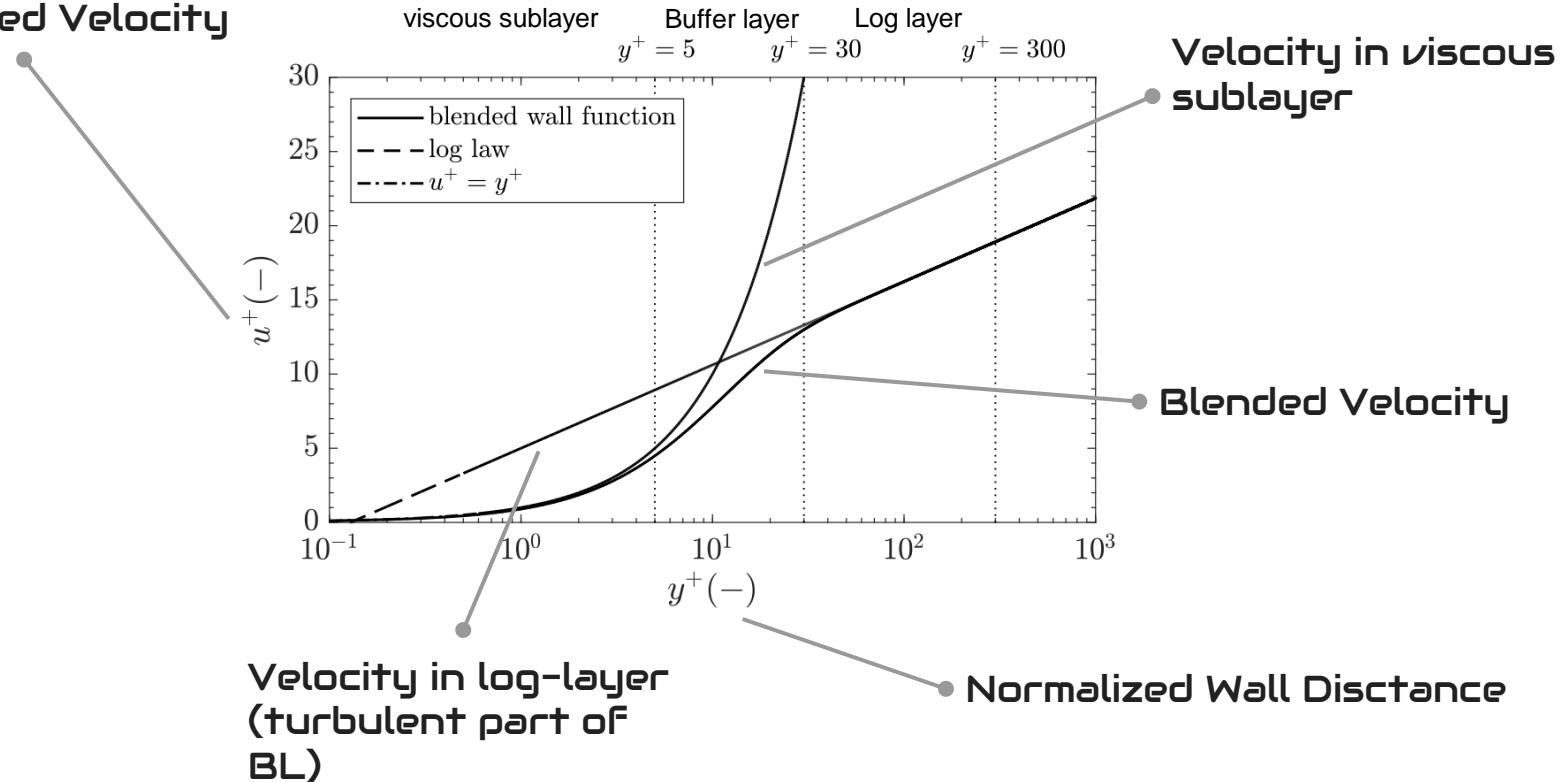
---

- » The aggregation of control volumes is known as **mesh** or **grid**. Discrete control volumes are named **cell** or **grid point**
- » Different mesh types are used for different flow characteristics
- » Regions with high gradients of physical quantities require finer discretization
- » High gradients to be found around geometries, shear layers and shock waves



# Meshing

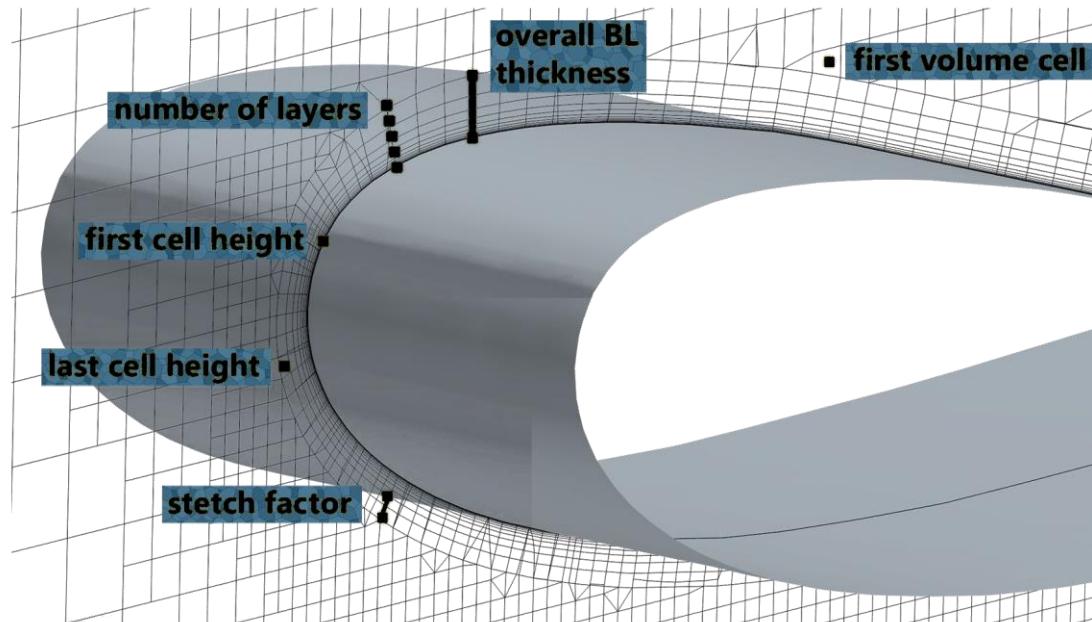
Normalized Velocity



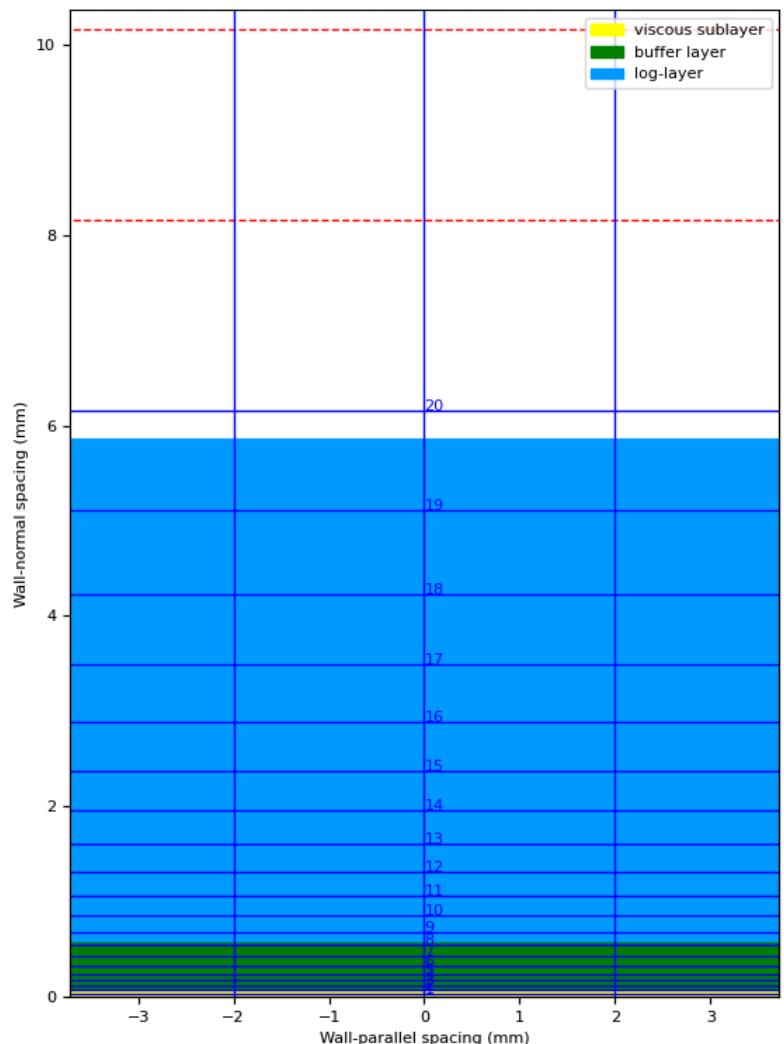
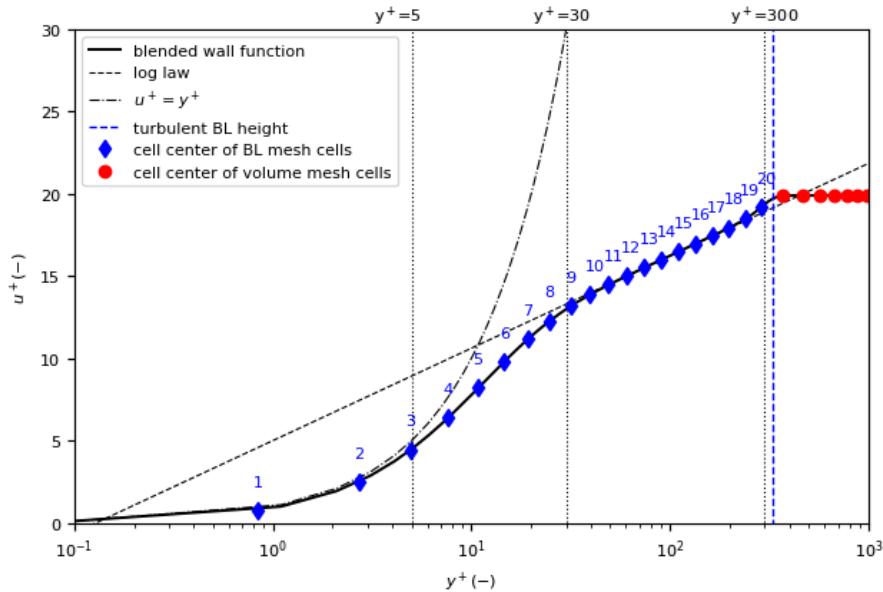
# Meshing

---

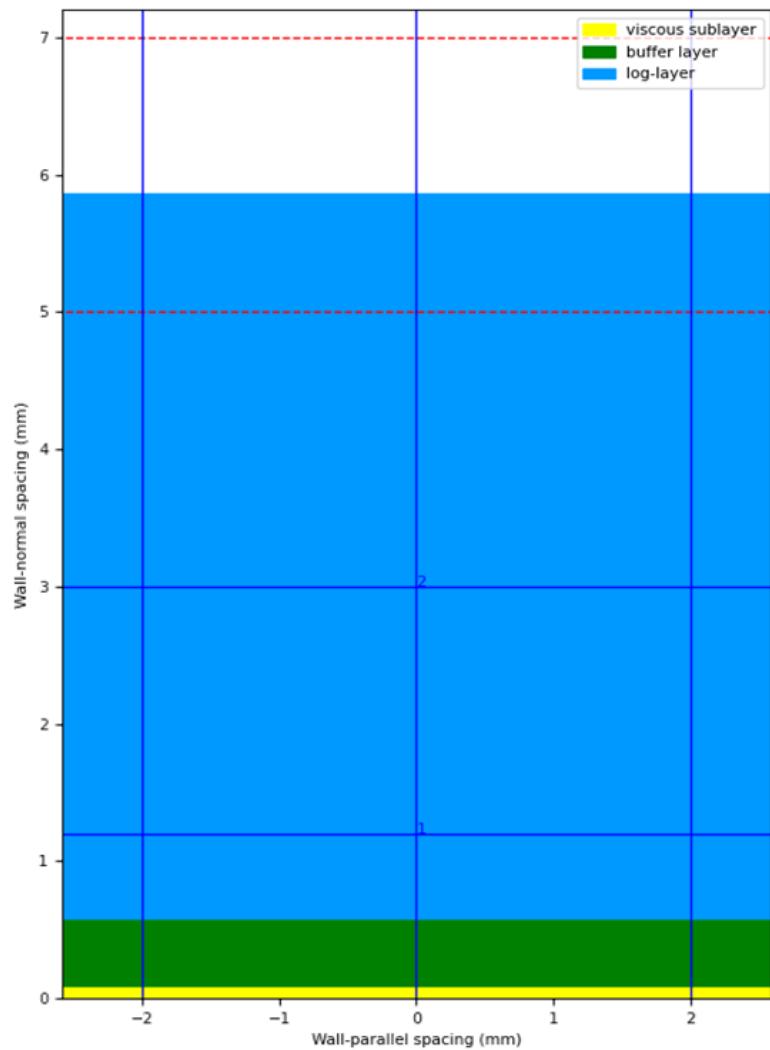
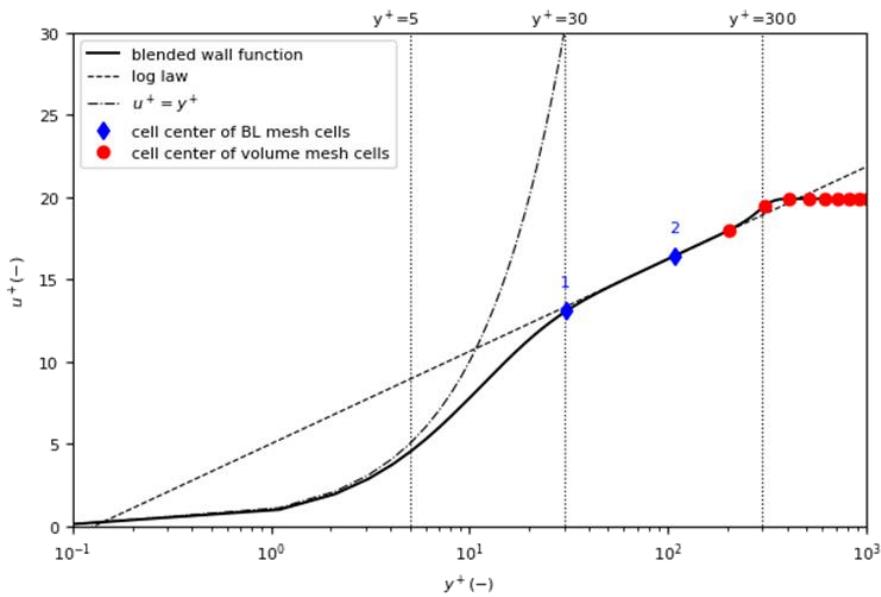
## Near Wall Meshing: Boundary Layer Mesh



# Meshing Low $y^+$ Mesh



# Meshing High $y^+$ Mesh



# Limitations and Errors

---

## Numerical

- Turbulence modelling
  - bluff body flows and highly separated flow
  - separation and reattachment points
  - high streamline curvature
  - transition modelling
- Discretization errors
  - mesh resolution
  - mesh quality
- Convergence
- Simplified Physics

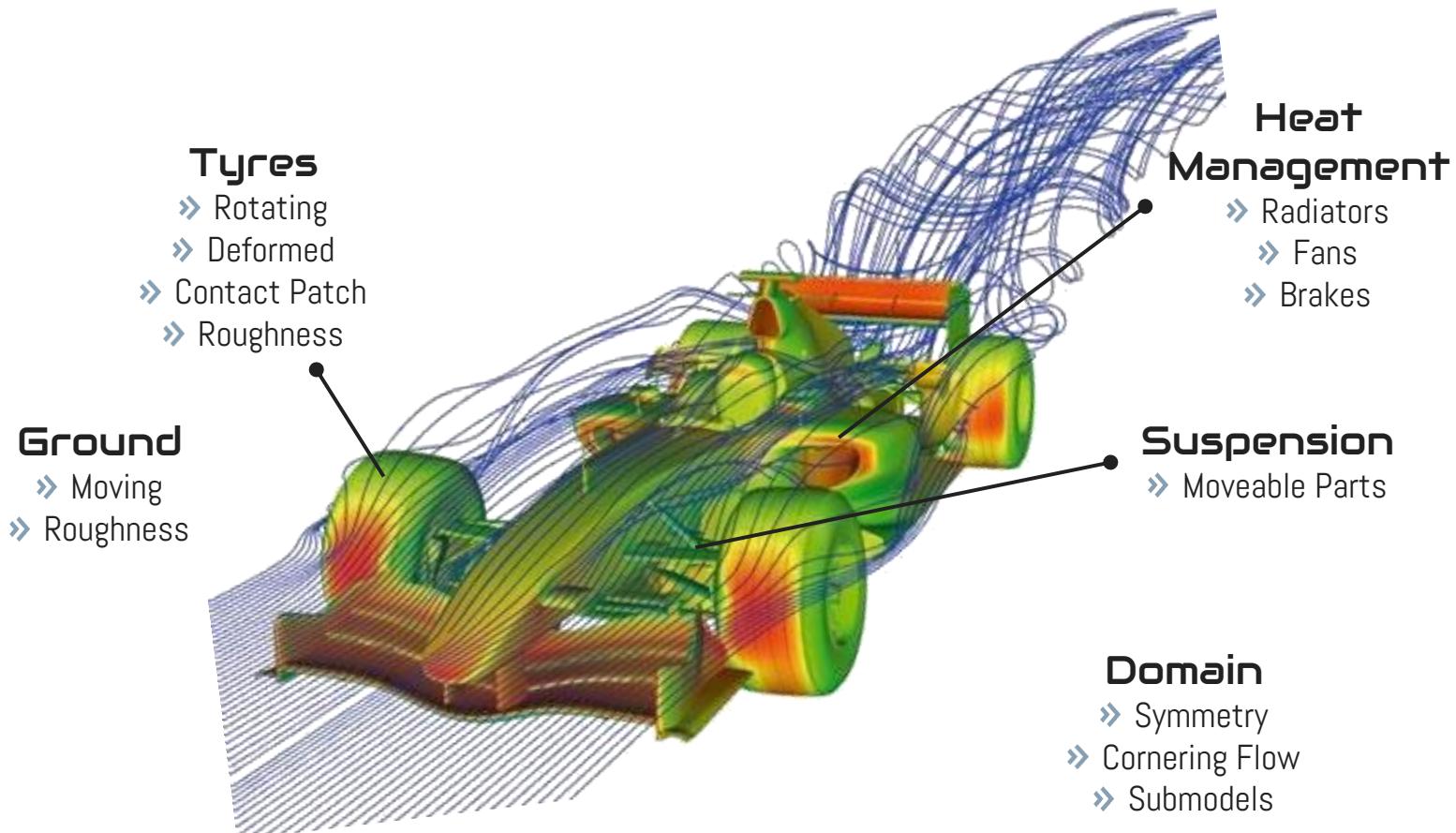
## Boundary Conditions and Geometry

- Roughness
- Inaccurate Geometry
- Simplified Geometry
- Wrong Assumptions for Boundary Conditions

2

# Race Car CFD

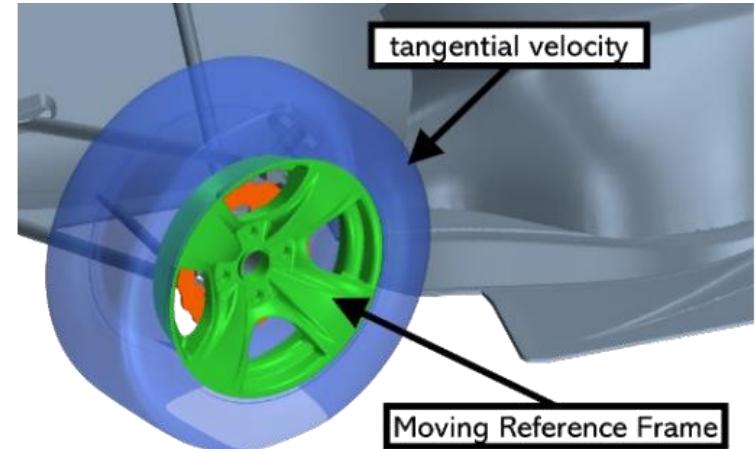
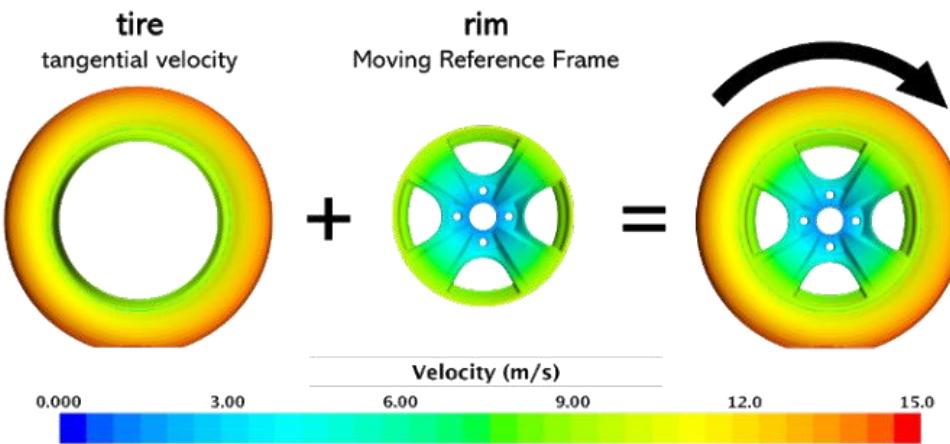
---



# Tyres

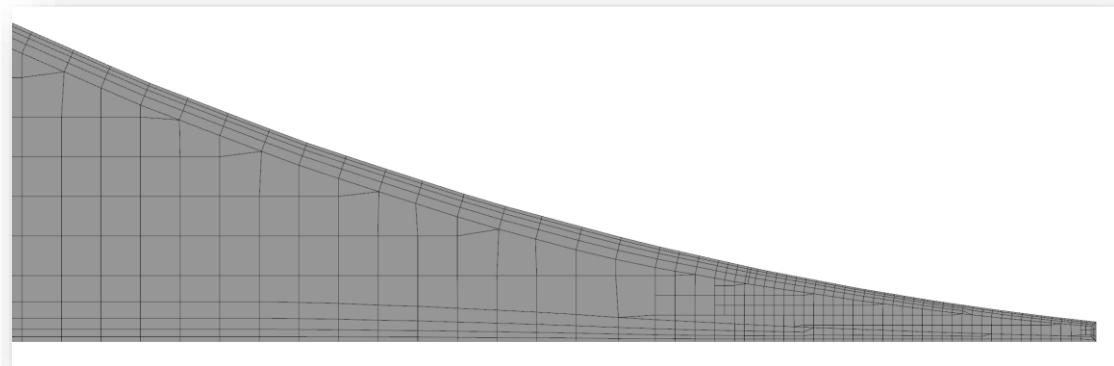
Moving Reference Frame (MRF):

- Modeling technique to simulate flows relative to a moving system
- Rotating/Translating Coordinate System where rotating components appear stationary relative to this frame
- Introduction of additional centrifugal and Coriolis forces



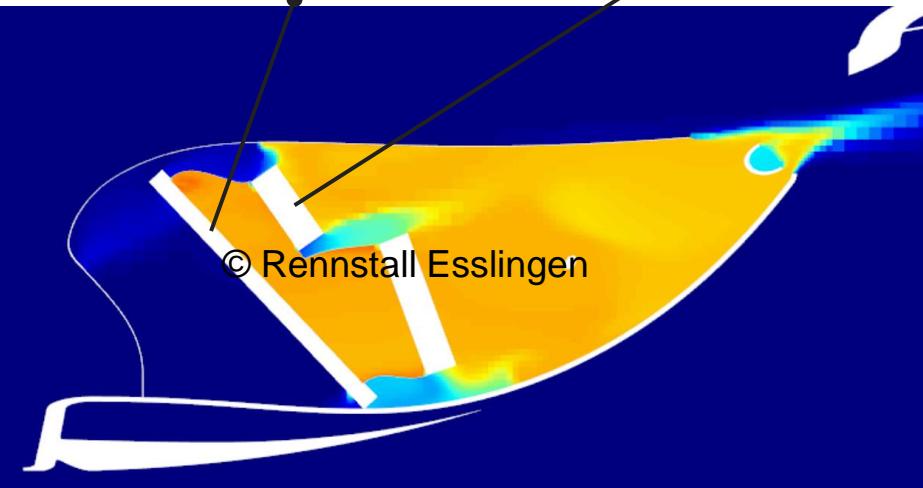
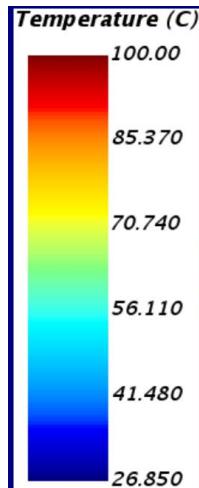
# Tyres

---



# Heat Management

- Radiators are modelled with a porous media: Artificial Resistance to model the pressure drop of the radiator
- Monitor flow uniformity and mass flow distribution on radiator surface
- You can apply a heat source to the radiator



## Porous Media / Porous Zone

- » Artificial Resistance in all 3 spatial directions

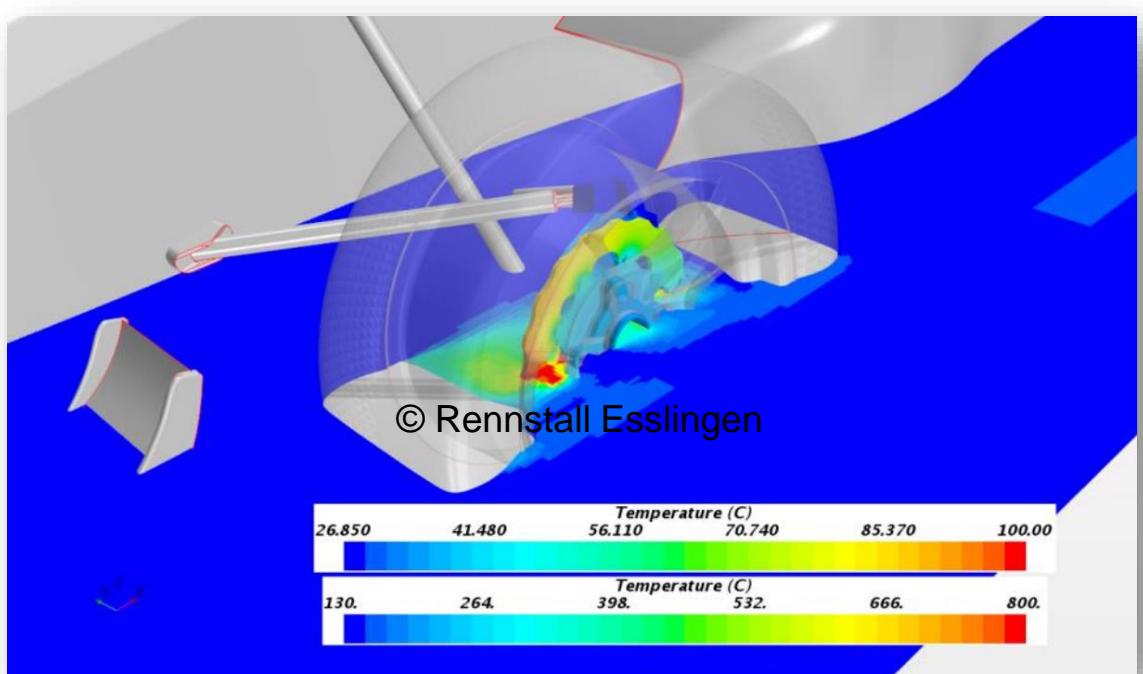
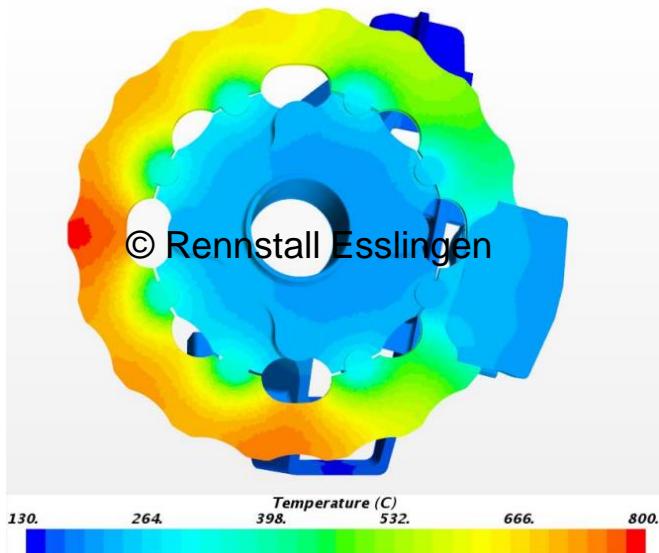
## Fans

- » Modelled with a „Fan Interface“ or MRF

# Heat Management

---

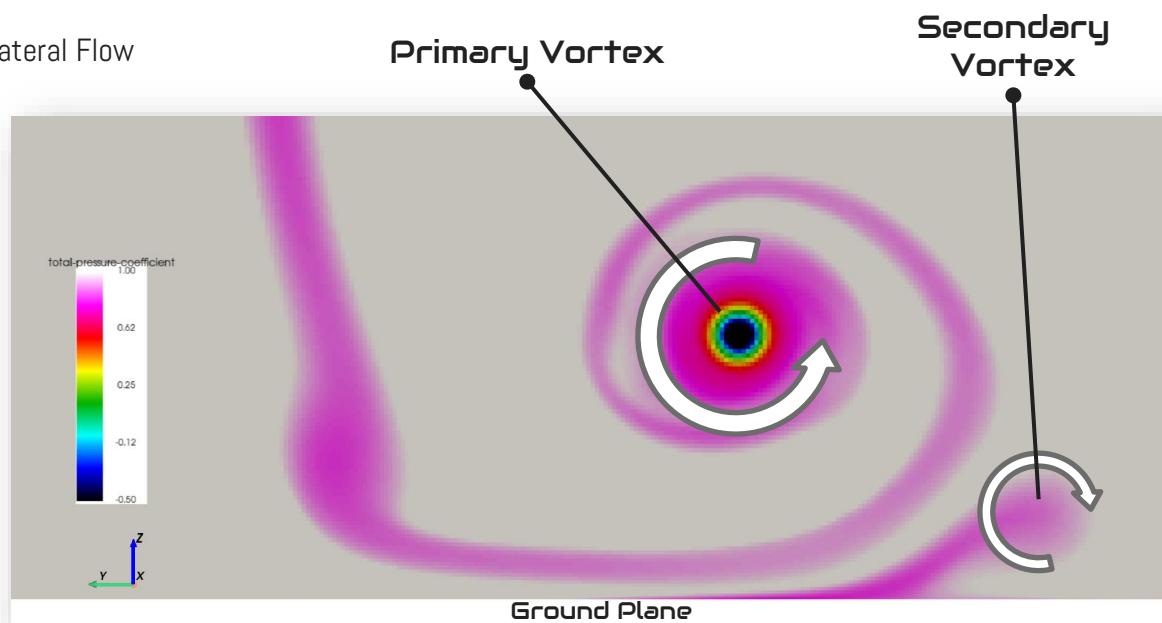
- Brake Cooling



# Ground

---

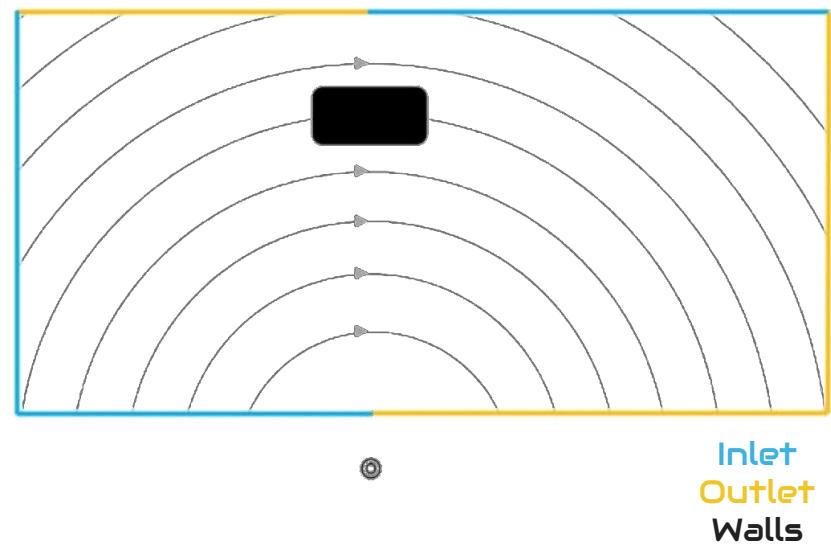
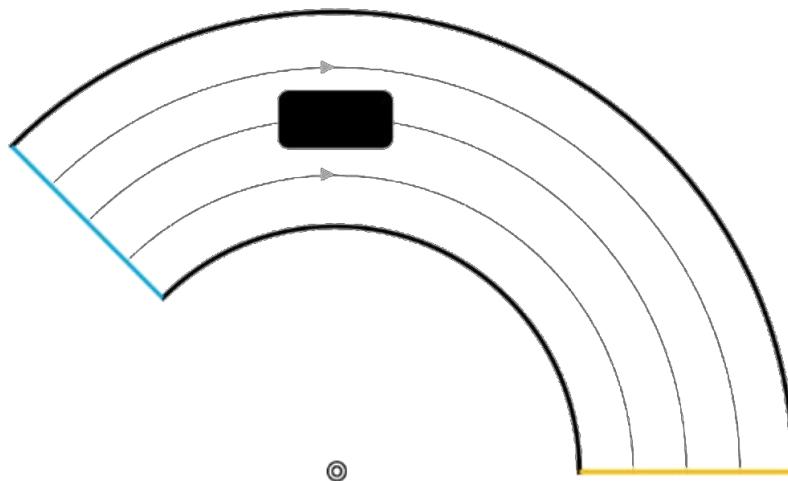
- Moving Wall
- Ground slip angle for Cornering
- Roughness
- Mesh
- Strong Interaction with Vortices and lateral Flow



# Domain

---

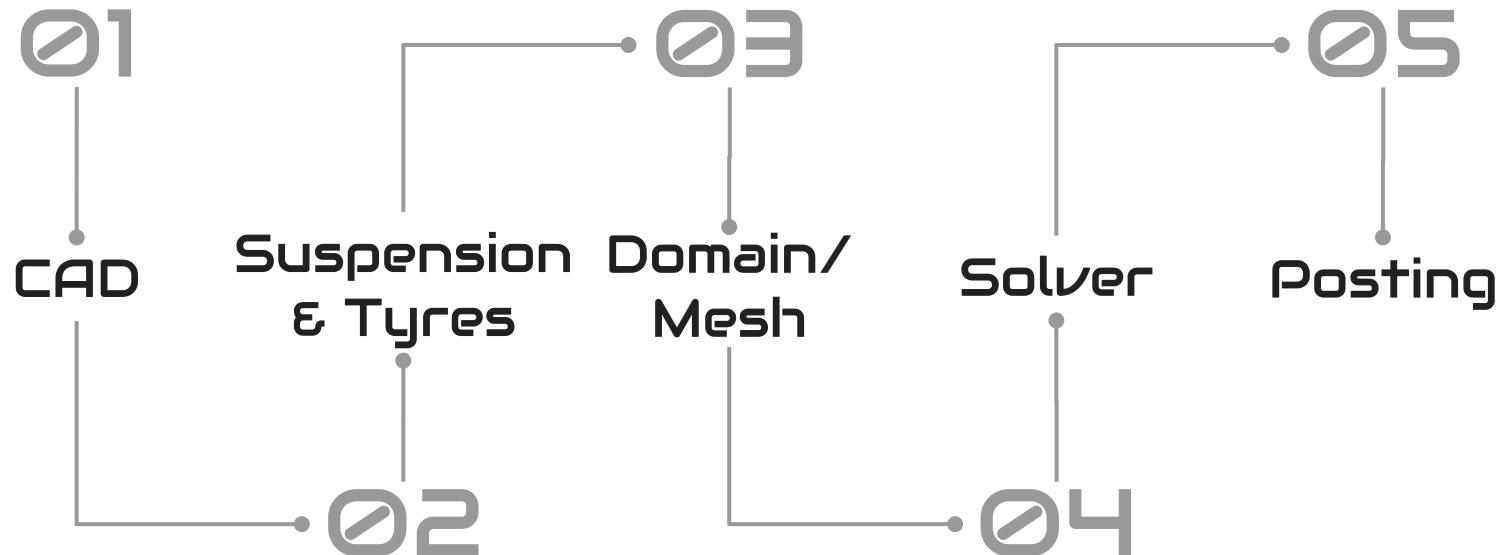
- Straight line: Rectangular Domain
- Curved Flow: Curved or Rectangular Domain with "Moving Reference Frame"





# A Workflow for Formula Student

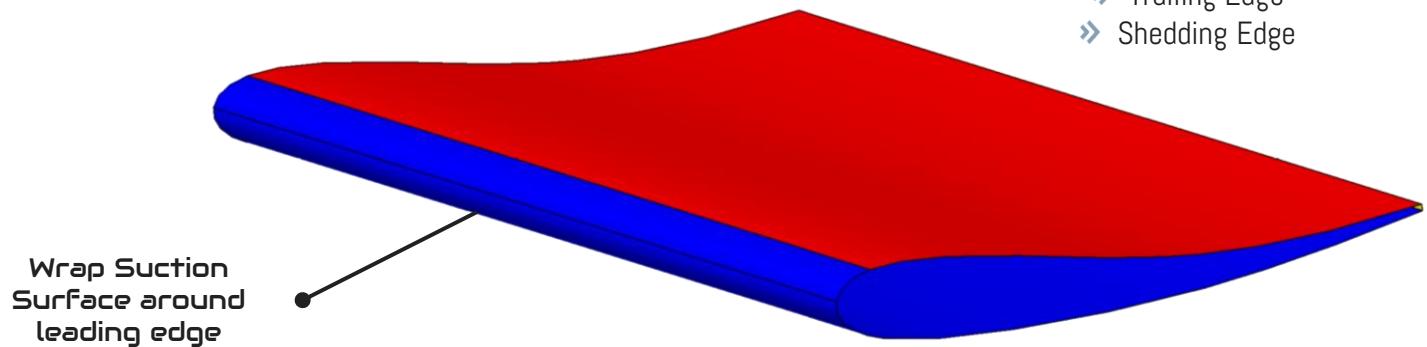
# Process Overview



# 01 CAD

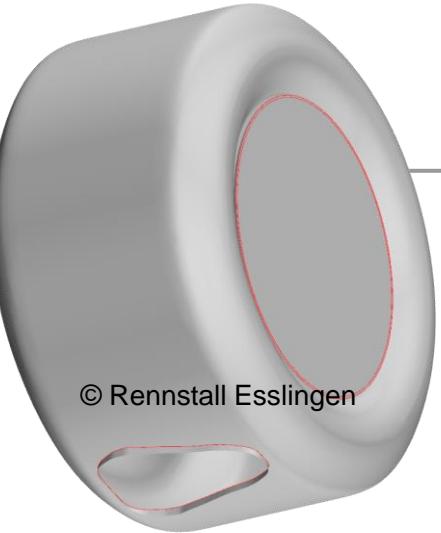
---

- Split Patches in CAD for efficient meshing strategies
- Create efficient Mesh Refinement Boxes in CAD
- Create Massflow-Planes in CAD
- Model the driver as accurate as possible for rear wing onset flow prediction

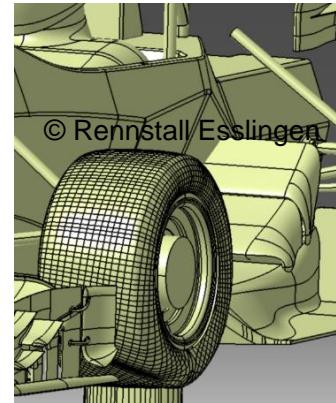


## Ø2 Suspension & Tyres

- Create (simple) functions to adjust the suspension
- Recreate the tyre contact-patch deformation
- Separation Point on tyre crucial for tyre wake prediction



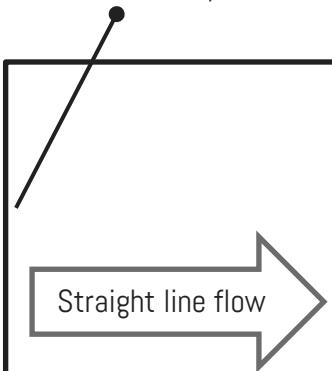
© Rennstall Esslingen



# 03.1 Domain

## Velocity Inlet

- » Fixed Velocity



Approx. 5 car lengths

## Sidewalls

- » Slip condition or Symmetry



## Pressure Outlet

- » Ambient Pressure



Min. 2-3 car lengths

## Height

- » Min. 4 car heights
- » More when strong upwash from wings

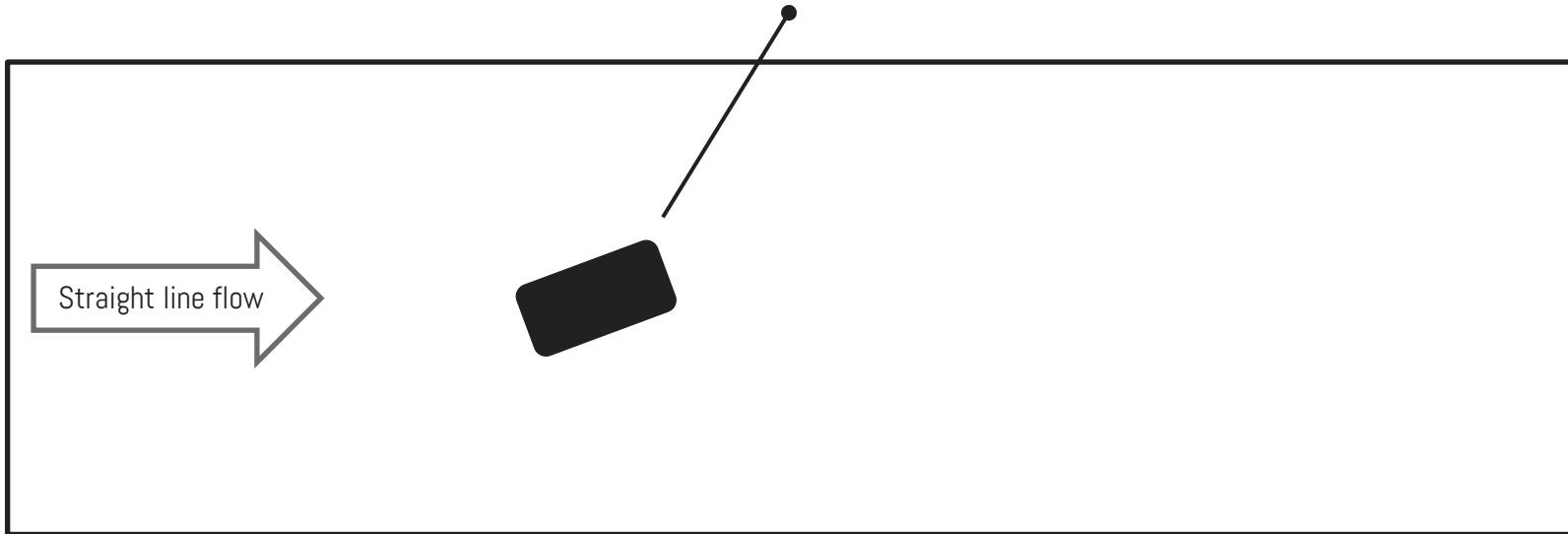
Approx. 10+ car lengths

# 03.1 Domain

---

## **Yaw Simulations**

- » Rotate Car for Yaw Simulations
- » This is not a cornering simulation !

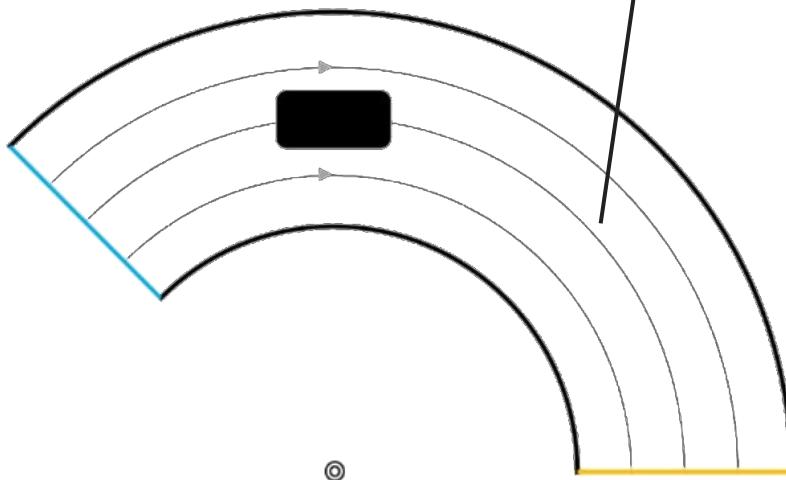


# 03.1 Domain

---

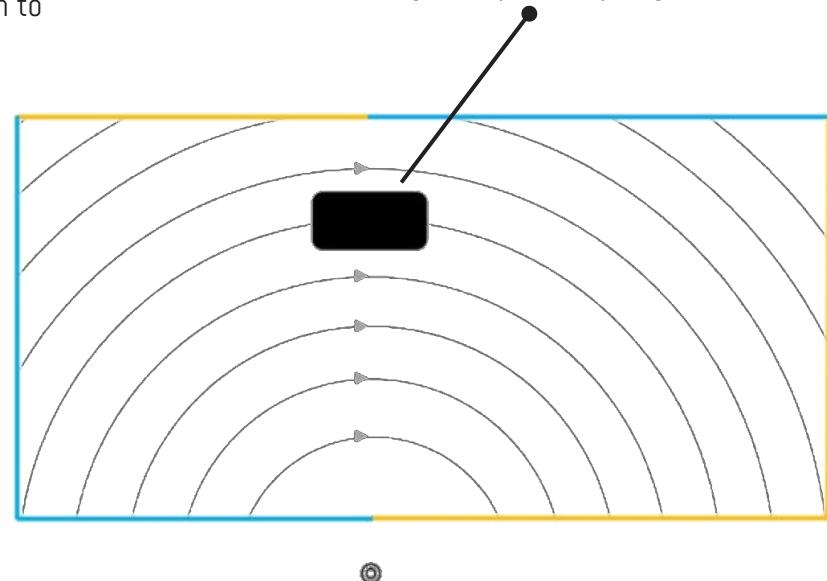
**Use a Moving Reference Frame to model the curved flow**

- » Change Inlet Boundary Condition to velocity = 0



**Rotate Car to account for Slip Angle**

- » Additional car rotation applies ground plane slip angle



## 03.2 Mesh

- Be smart with your wall-treatment and refinements
- Use Boxes for refining the mesh where you need it
- Offset Surfaces or Distance Refinements for Geometry related meshing
- Mesh Study

Use Curvature  
Refinements

Suction Surface

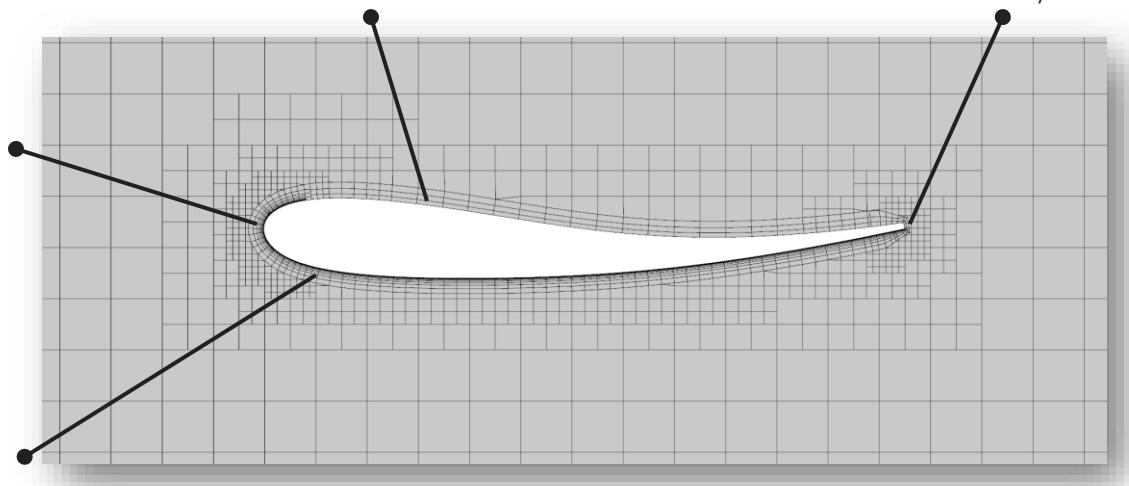
- » Low yPlus
- » Fine mesh
- » Distance refinement

Pressure Surface

- » High yPlus
- » Coarser mesh

Trailing Edge

- » High yPlus
- » Fine mesh
- » Few layers



## 03.2 Mesh

---

### Mesh Study

- Perform wall-based and volume-based mesh studies
- Wall-base: low and high  $yPlus$ , number layers, thickness of layered mesh and thickness of first cell height
- Investigate every flow structure separately
- Get a feeling where the critical and most sensitive areas are
- Do not just compare loads, compare the flow field and skin friction

# 04 Solver

---

## **First Choice: Coupled Solver**

- » Usually faster than Segregated Solver
- » Higher Memory Requirements
- » Try decreasing the computational effort for solving by keeping the simulation stable:  
Under Relaxation, Pre- and Postsweeps,  
Number of Iterations for the linear Solver,  
Cycle type, CFL number

## **Alternative: Segregated Solver**

- » Use it when you get memory exceptions with the coupled solver
- » If you have convergence issues with coupled solver
- » Coupled solver probably not available in your CFD code

# 04 Solver

---

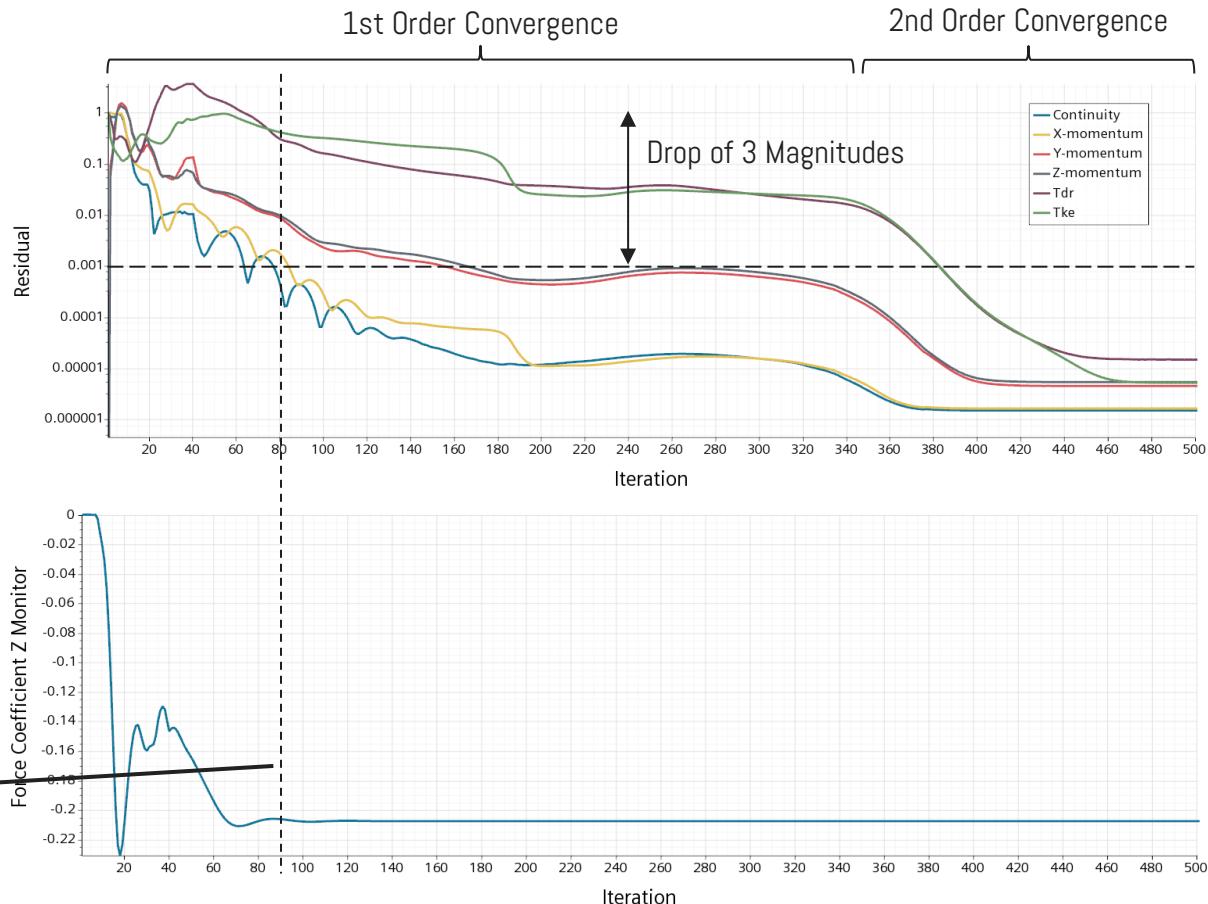
## Solutions for limited computational resources

- Only RANS simulations, transient does not make sense for development
- Try optimizing your mesh to reduce cell count by maintaining a good level of accuracy
  - Formula Student CFD should have at least 40-60mio Cells Fullcar
  - If that is not possible, you can still use your CFD but do not assume the loads and flow field is fully correct
- Use Sub-Models for Development:
  - Frontwing + Front Wheel + Front Chassis
  - Rear Models with mapped Inlet conditions
  - You can also use models with a coarser mesh at the front/rear but be aware of inaccuracies
- Do not use 2D simulations!

# 04 Solver

## Convergence

- Residuals: Difference between current and last iteration
- Rule of thumb: Residuals should drop by 3 Magnitudes
- Monitor Forces, not only residuals



Steady State For  
Force reached

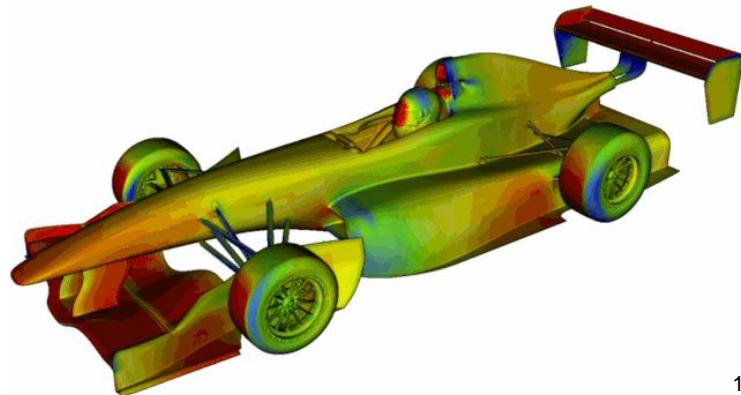
# 05 Postprocessing

---

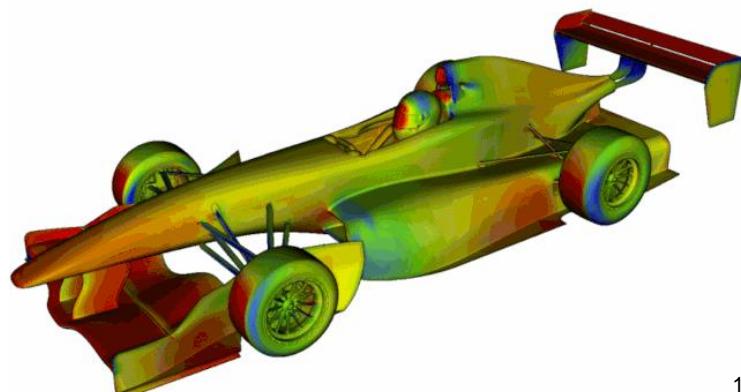
## General

- Use a coordinate System that is steady relative to the unsprung car
- Normalize all variables and forces
- Use the right colorbars and fixed limits
- Use massflow planes to monitor the massflow through brakes, radiators, ...
- You can also use massflow planes to monitor the total pressure average in some areas
- Calculate aero balance
- Output forces, massflow, balance, ... in a way that you can easily compare them

Postprocessing should be automated and repeatable!



1



1

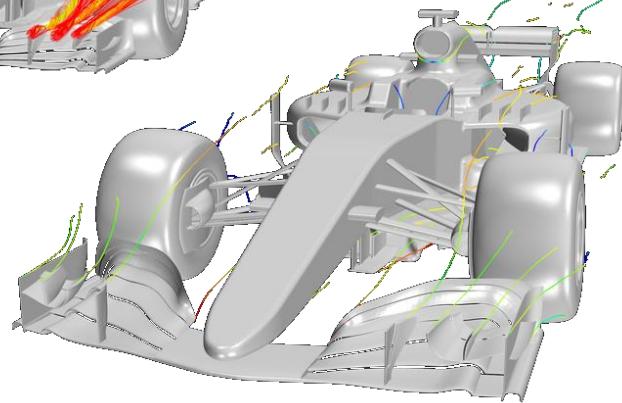
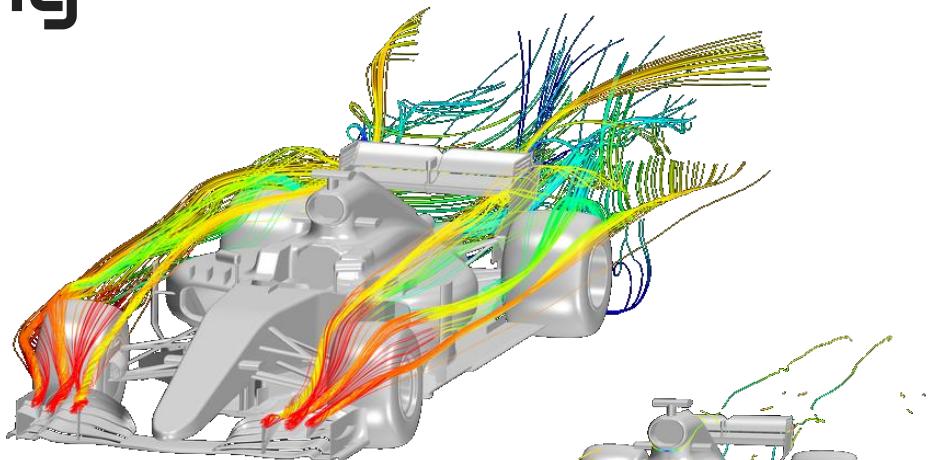
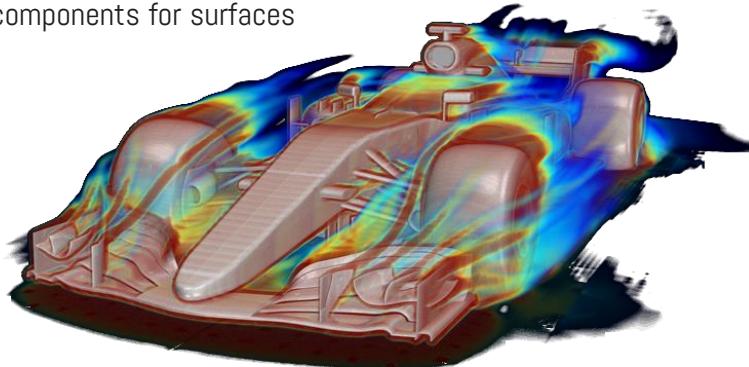
# 05 Postprocessing

---

## Flow Field Visualization

### 1. Understand the flow field

- Streamlines
- Vortex Cores
- Iso-Surfaces
  - Total Pressure
  - Vortex Criterion: Helicity/ Q or Lambda2
- Line Integral Convolution (LIC)
  - Velocity components for cutting planes
  - Skin Friction components for surfaces



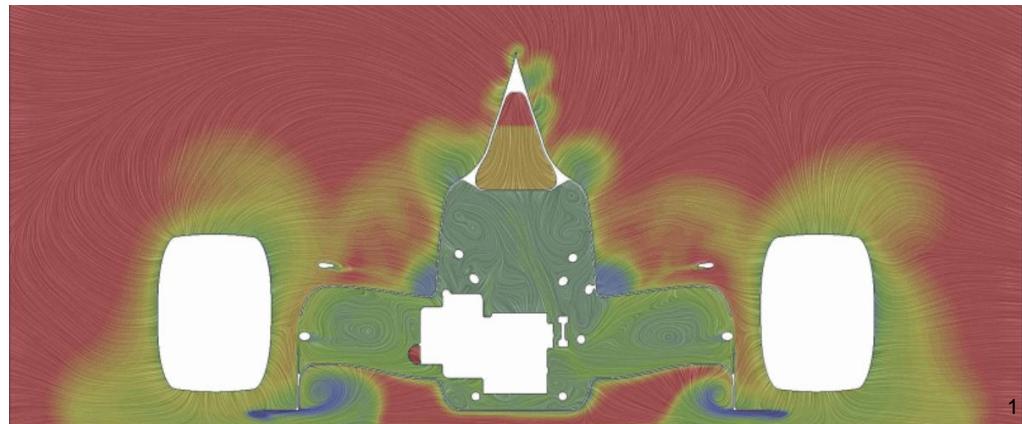
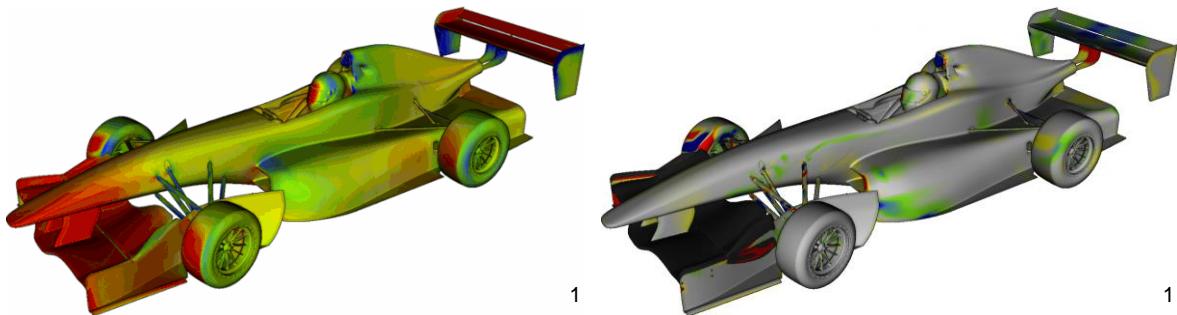
# 05 Postprocessing

---

## Flow Field Visualization

### 2. To compare design iterations

- Cutting Planes
  - Total Pressure
  - Pressure
  - Vortex Criterion
  - Velocity (+ Components)
- Surface Variables
  - Pressure
  - Skin Friction
- Delta Calculation
- Surface Slice Plots
  - Pressure
  - Skin Friction



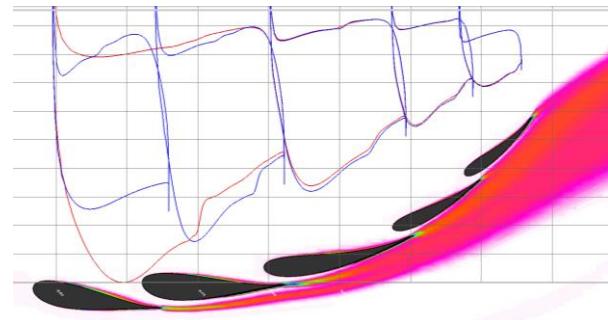
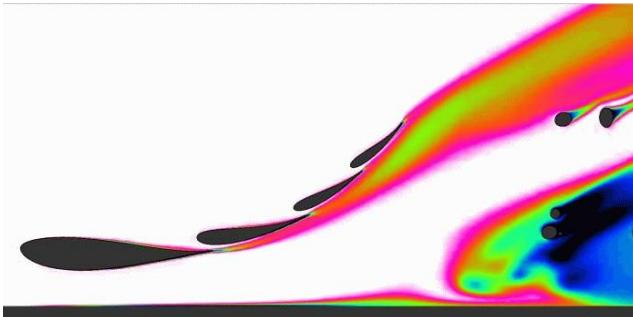
# 05 Postprocessing

---

## Flow Field Visualization

### 2. To compare design iterations

- Cutting Planes
  - Total Pressure
  - Pressure
  - Vortex Criterion
  - Velocity (+ Components)
- Surface Variables
  - Pressure
  - Skin Friction
- Delta Calculation
- Surface Slice Plots
  - Pressure
  - Skin Friction



4

Validation  
and  
Correlation

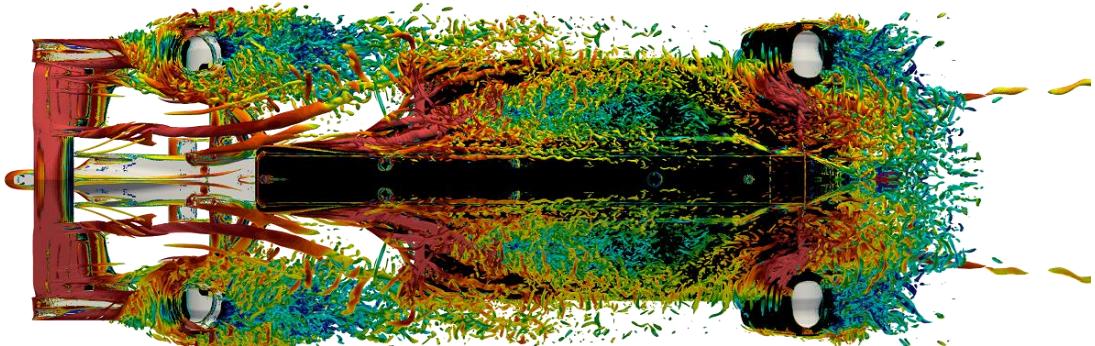
---

# Data Collection

---

Where to get your data to correlate to from?

- » Track testing
- » Windtunnel
  - Wool tufts
  - Flow viz paint
  - Pressure taps
  - Aero Rakes
  - Load measurement
- » High Fidelity Simulations
  - Volume and surface field data
  - Loads



# Data Collection From CFD

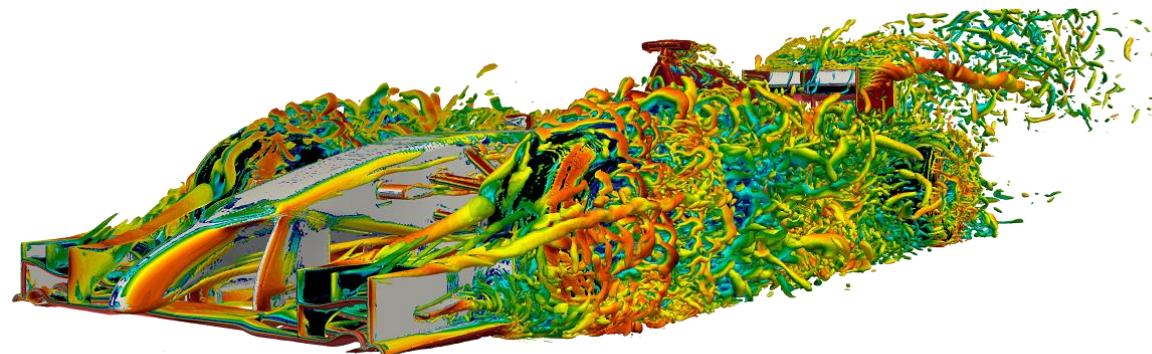
---

Turbulence Modelling one of the main issues in RANS

- We can use high fidelity CFD to get an idea of how close the RANS flow field is to the resolved turbulence
- This is not a validation of the whole simulation, but good to highlight some issues and the understanding of the RANS limits

Best to compare:

- » Wheel wakes
- » Vortex Positions, Vortex Strength and possible Bursts
- » Loads



# Adress the Problems

---

## Separation

Wing separates or separation point different from CFD

- » Check Geometry
- » Check mesh
- » Check onset flow
- » Is it laminar separation?
- » Check turbulence model
- » Investigate model parameters

## Different Flow Field

Wakes and losses in different position

- » Check Geometry
- » Is there a different separation behaviour upstream?
- » Mesh sensitivity
- » Does it look less or more dissipated? → try other turbulence model or adjust turbulence model
- » Investigate surface roughness

## Different Vortex Behaviour

Vortices appear stronger/weaker, burst or in different position

- » Check Geometry
- » Mesh sensitivity
- » Check Curvature Correction is activated and check parameters
- » Adjust turbulence model parameters

5

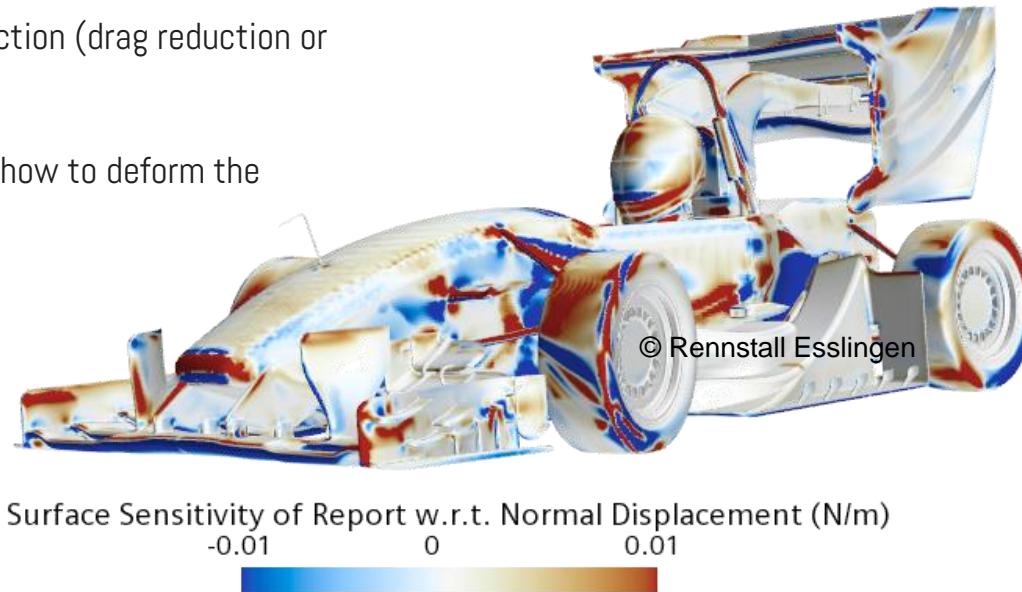
# Future Trends in CFD

---

# Adjoint

---

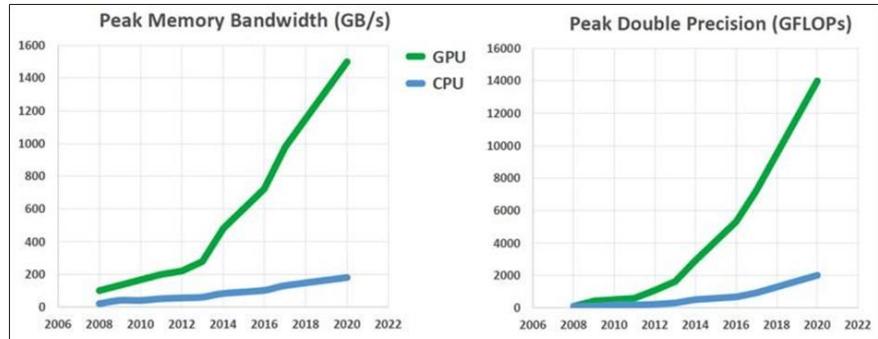
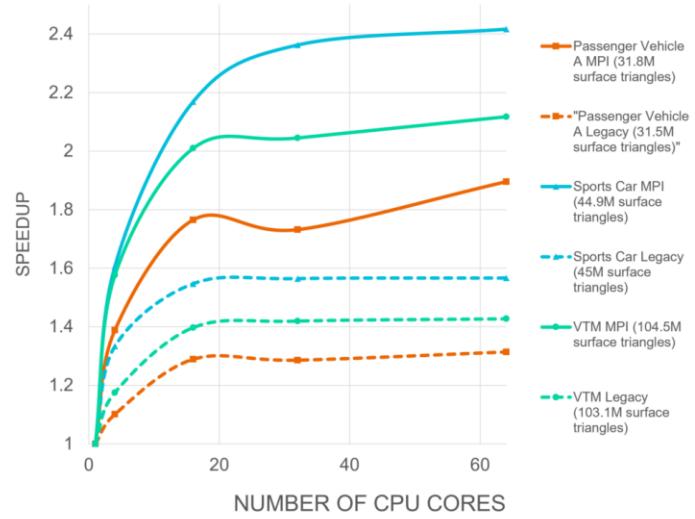
- Equations derived from the Navier-Stokes Equations
  - Fixed flow field to determine sensitivity gradients of design variables (mostly shape of geometry)
- » Allows for optimization of an objective function (drag reduction or downforce generation)
- » Can be applied in an optimization cycle
- » Sensitivity result will give an indication of how to deform the geometry
- » Limitation: Linear behaviour



# GPU Based Solving

**CPU (Central Processing Unit):** Optimized for sequential processing with a few powerful cores  
**GPU (Graphics Processing Unit):** Designed for parallel processing with thousands of smaller cores

- » Matrix operations in linear algebra are highly parallelizable
- » Bandwidth of communication between CPUs slows down the computation for simulations with many CPUs (non-linear scalability of number of CPUs to solve time)
- » GPUs can deliver significant speedups over CPUs, especially for large and transient simulations
- » GPUs still show an extreme development trend compared to CPUs

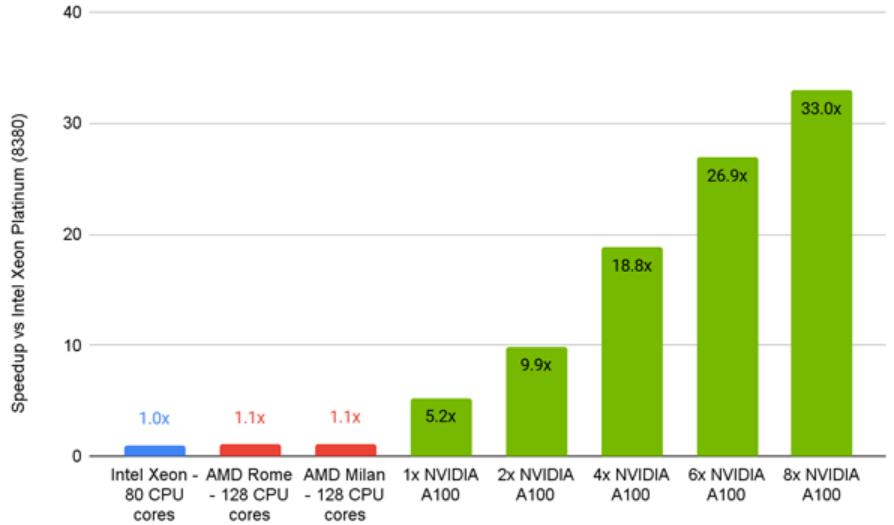


# GPU Based Solving

## Limitations

- » GPUs often have less memory than CPUs
- » not all software supports GPU solving
- » Availability and Cost of computing resources  
(GPUs also very attractive for Machine Learning)
- » Meshing better suited for CPU processing

The future will likely be a hybrid CPU-GPU setup to leverage the strength of both setups



# Machine Learning

---

Machine Learning is a branch of artificial intelligence that enables systems to learn patterns from data and make predictions or decisions without being explicitly programmed.

**Machine Learning in CFD used to:**

- Reduce computational cost
- Enhance Turbulence Modelling
- Analysis of Simulations
- Assisting in Setup work and automate Workflows
- Improve accuracy

**Challenges:**

- Model needs to be trained with lots of data
- Interpolation better performed than extrapolation
- Extensive Validation required
- Ensuring Physics

# Machine Learning

---

## Surrogate Modelling

- » Predicting the Solution without solving the NS-Equations
- » Trained with data from previous simulations
- » Predictions made by interpolating/extrapolating from the training dataset

## Accelerating Solvers

- » Initialize the simulation with an assumption from the ML model
- » Convergence acceleration by solver adjustments from the model

## Turbulence Modelling

- » Enhance existing TM by learning from high-fidelity data
- » Adjustment of TM for different flow regimes and characteristics
- » Up to full replacement of TM with AI Model

## Workflow and Setup Assistance

- » ML to help setting up CFD cases
- » Assisting in mesh generation
- » Adaptive mesh refinements

# Join the Audi F1 Project

## Graduate Opportunities

### Who We're Looking For:

We're offering entry-level positions for recent graduates across various departments:

- Engineering
- Aerodynamics
- Design
- Data Analysis
- Business Operations

### Your profile:

- A Bachelor's or Master's degree in a relevant field.
- A genuine love for motorsport and a desire to be part of Formula 1.
- Strong analytical, problem-solving, and teamwork abilities.
- Excellent communication skills and attention to detail.
- Adaptability and a willingness to learn in a fast-paced, ever-evolving environment.
- Relevant internship or experience in the motorsport or automotive industries is a plus, but not essential.
- Full professional proficiency in English and a willingness to learn basic German.

## Internships

Join us on a full-time internship over 6 month or up to a year!

Check our website for open position.

## Trainee Program

Will be launched in 2025. Rotate through various engineering departments within the team over 24 months and find the position that fits you the best!



Audi F1 Project

[sauber-group.com/corporate/jobs](http://sauber-group.com/corporate/jobs)

