

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, ...

2 Race Car CFD

Key Aspects of Modelling Race
Car Components

3 A Workflow For Formula Student

All Steps from CAD to
Postprocessing

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

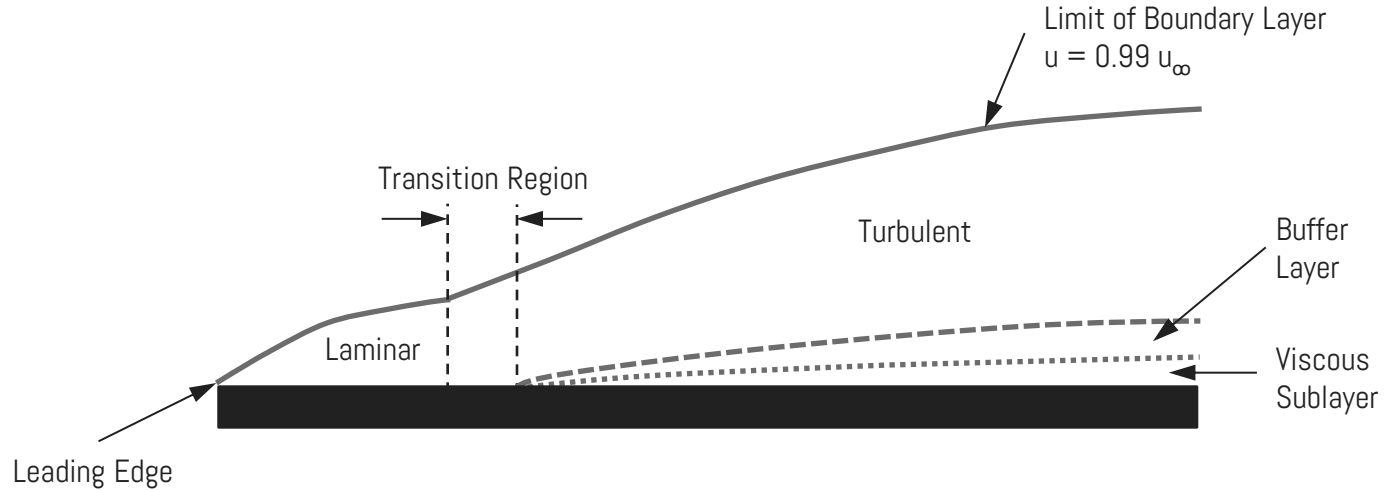
Radical Design Exploration

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

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

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

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

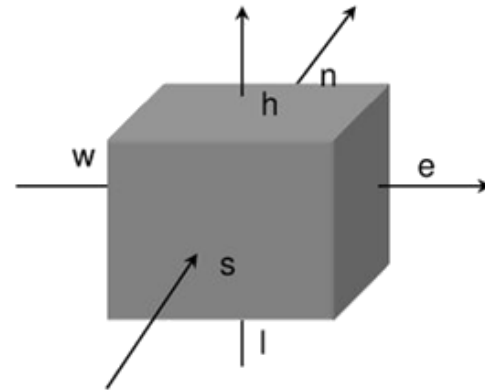
- » Numerical Techniques such as **Finite Volume Method** (FVM) are used to compute the quantities in each control volume
- » The continuous space is described by a finite set of equations that can be handled by the computer
- » The finer the discretization, the more accurate the solution

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



- 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 , ...

» pressure-velocity link and nonlinear terms do not allow to solve the system in one step

» Iterative Solving Process

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

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

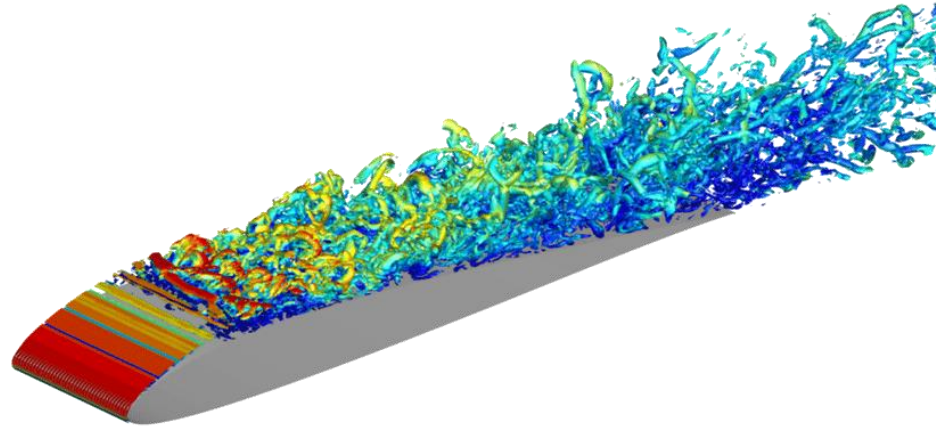
Possible to solve Navier-Stokes equations directly, but it requires:

- 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

RANS

Time-resolved
Resolved time-dependant flow structures

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

High computational cost

Low computational cost

Turbulence resolved or minimally
modelled

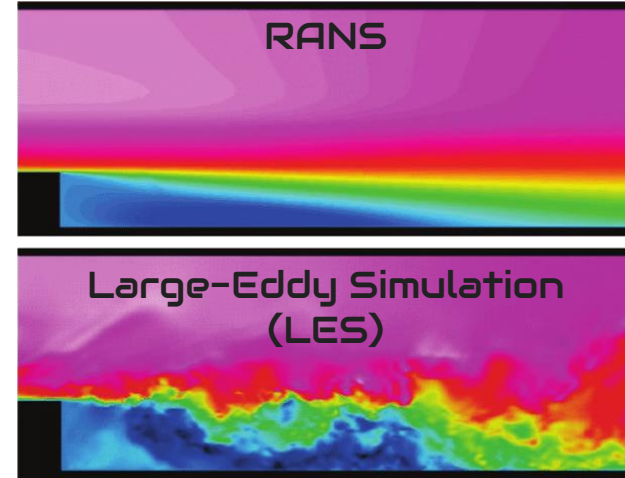
Modelled Turbulence

Best for dynamic flows

Best for steady-state flows

Needs a very fine mesh

Coarser meshes possible



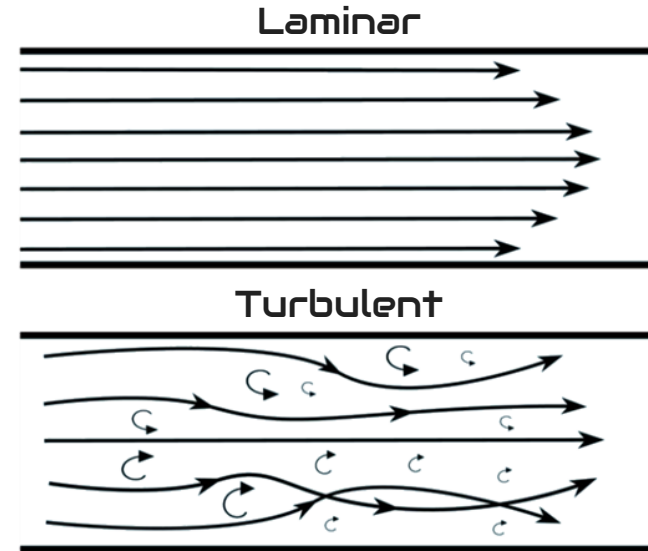
Turbulence Modelling

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

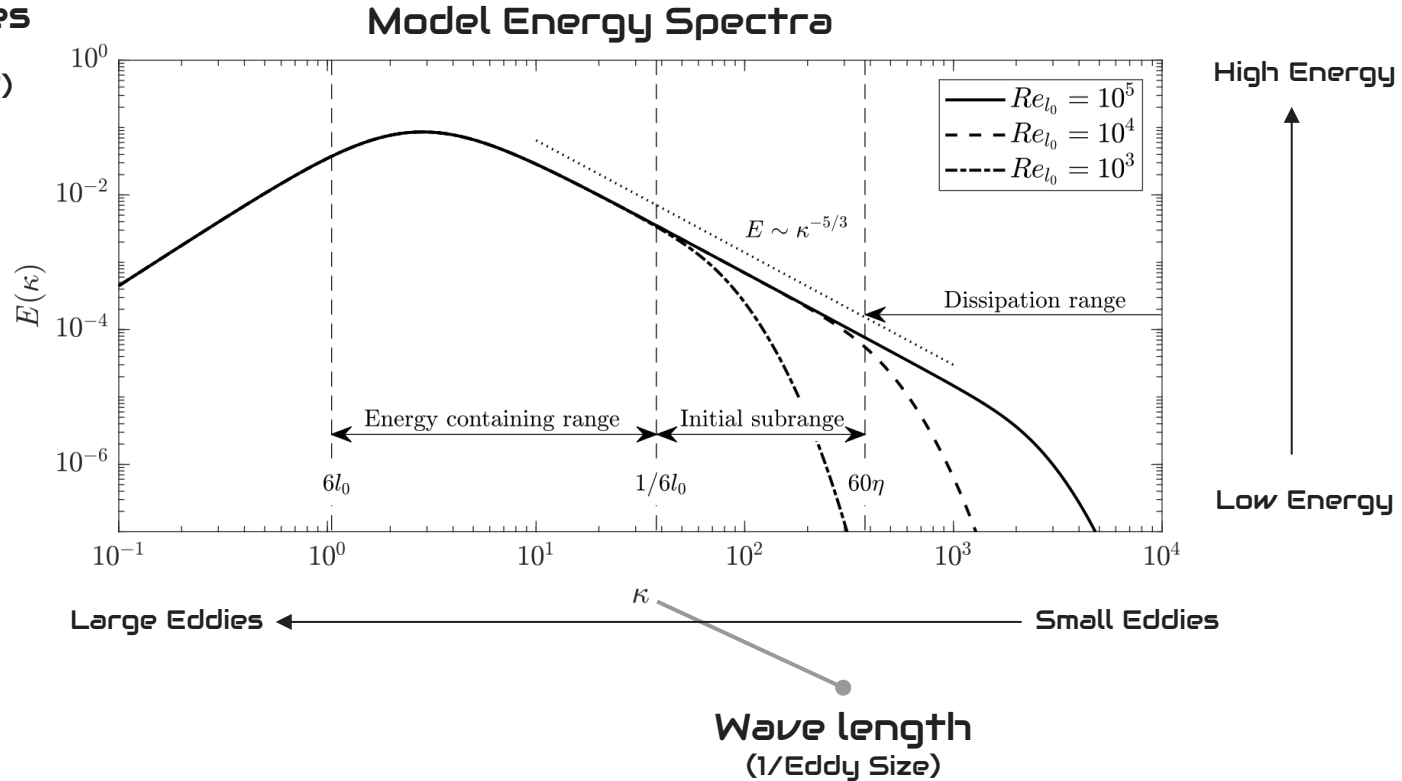
How can the turbulent viscosity be calculated?

- 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

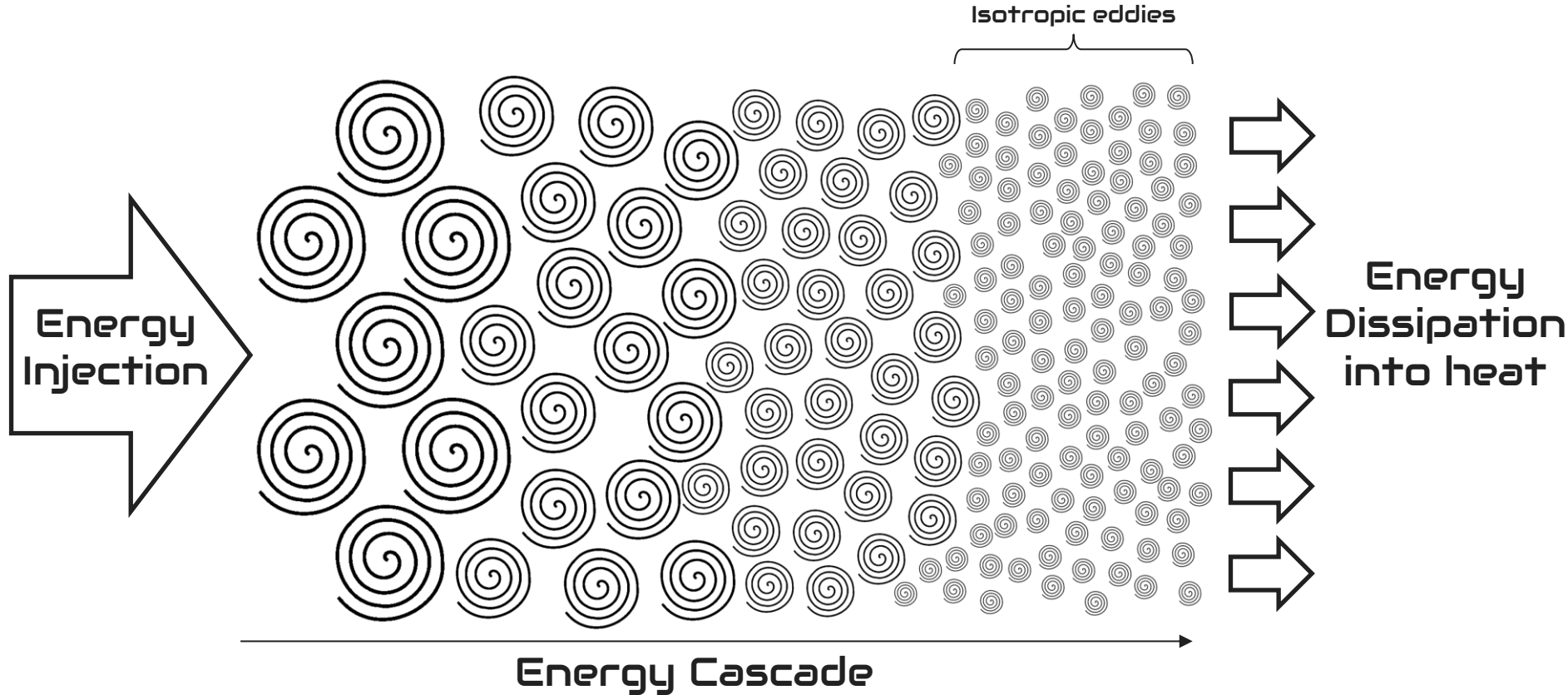


Turbulence Modelling

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



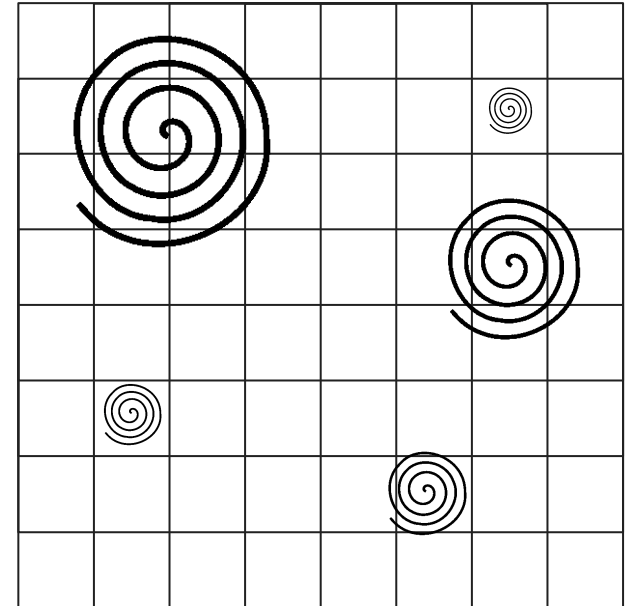
Turbulence Modelling



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**

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

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

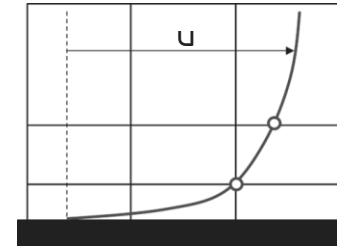
k: turbulent kinetic energy

ϵ : turbulent dissipation rate

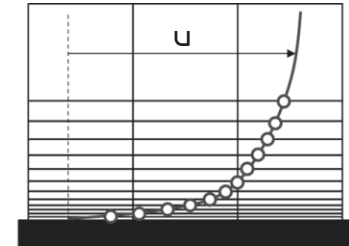
ω : specific rate of dissipation

- » 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

Wall Treatment



High y^+
Using wall Functions



Low y^+
Resolved Flow

Turbulence Modelling

	Realizable k-epsilon	SST Menter k-omega
Formulation	Improved and more physically accurate than standard k-epsilon	Blends between k-omega near walls and k-epsilon in far field
Wall Treatment	Needs additional models for low y^+ *	All y^+
Transition	No**	No**
Adverse Pressure Gradients	Moderate	Excellent
Free-shear Flows	Good	Moderate
Separated Flows	Poor	Excellent
Application	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

Disadvantages/Problems

Steady-State (RANS)

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

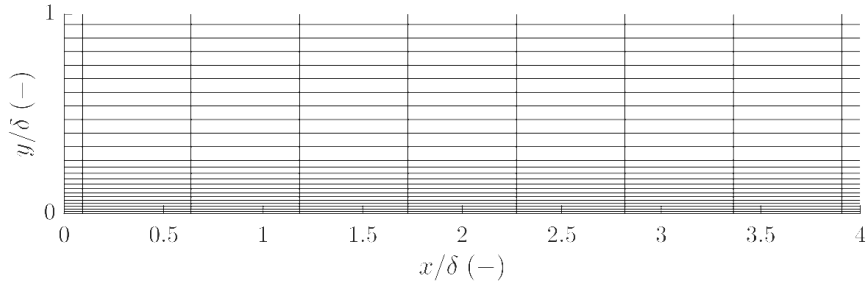
- » Low accuracy for transient phenomena
- » Limited in separated flows/high streamline curvature
- » Empirical dependency

Transient (LES)

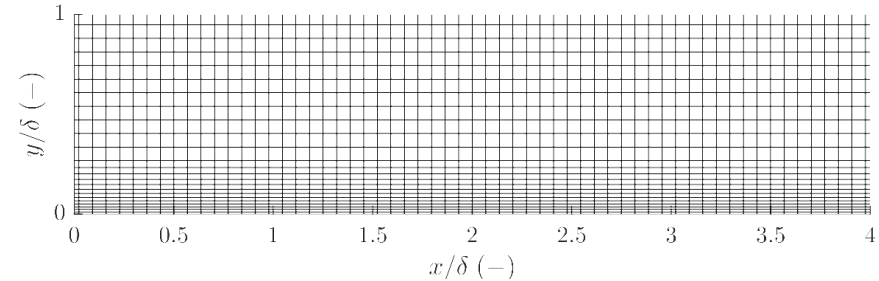
- » Captures detailed turbulent structures
- » Great for unsteady and complex flows

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

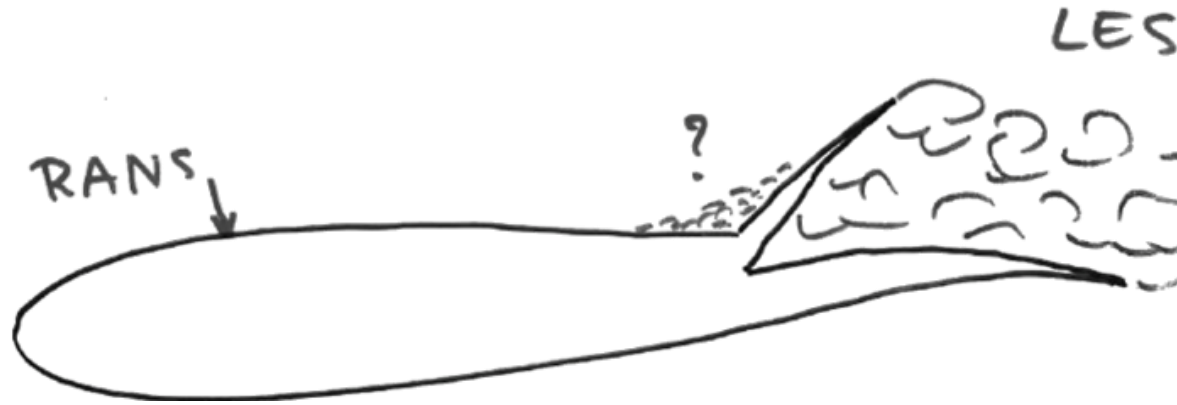
RANS near-wall Mesh



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)

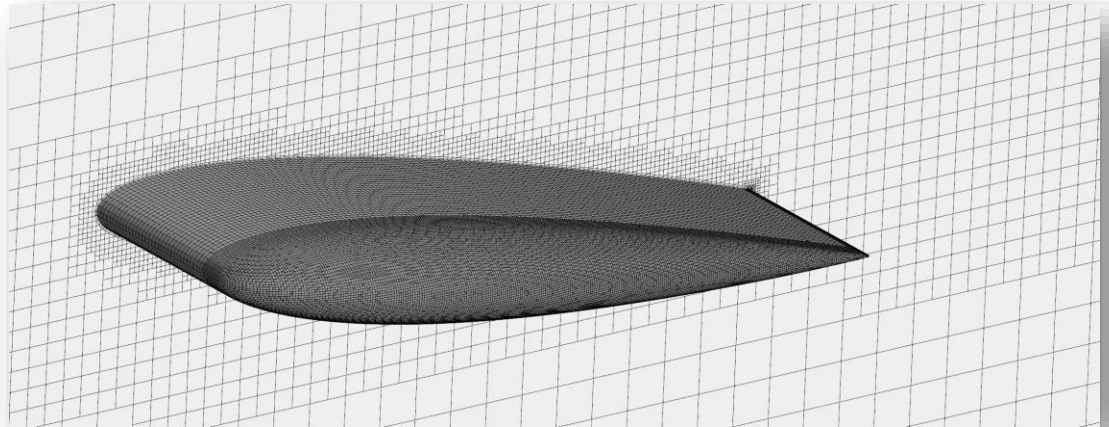
- Ensures the boundary layer is always treated by RANS and the interface to the LES region lies outside of the boundary layer
- Should always be used instead of the original DES
- » Boundary Layers are modelled with RANS limitations

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

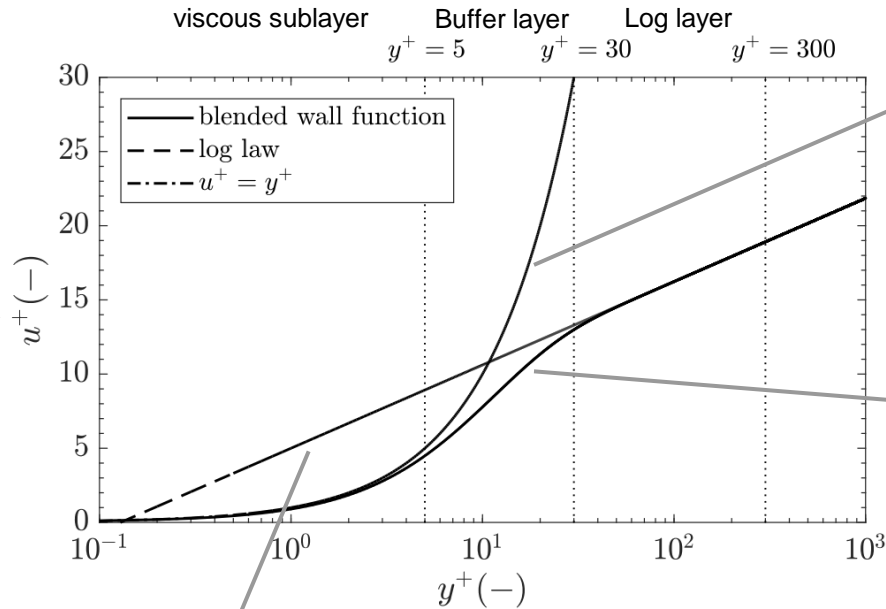
Meshing

- » 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



Velocity in viscous sublayer

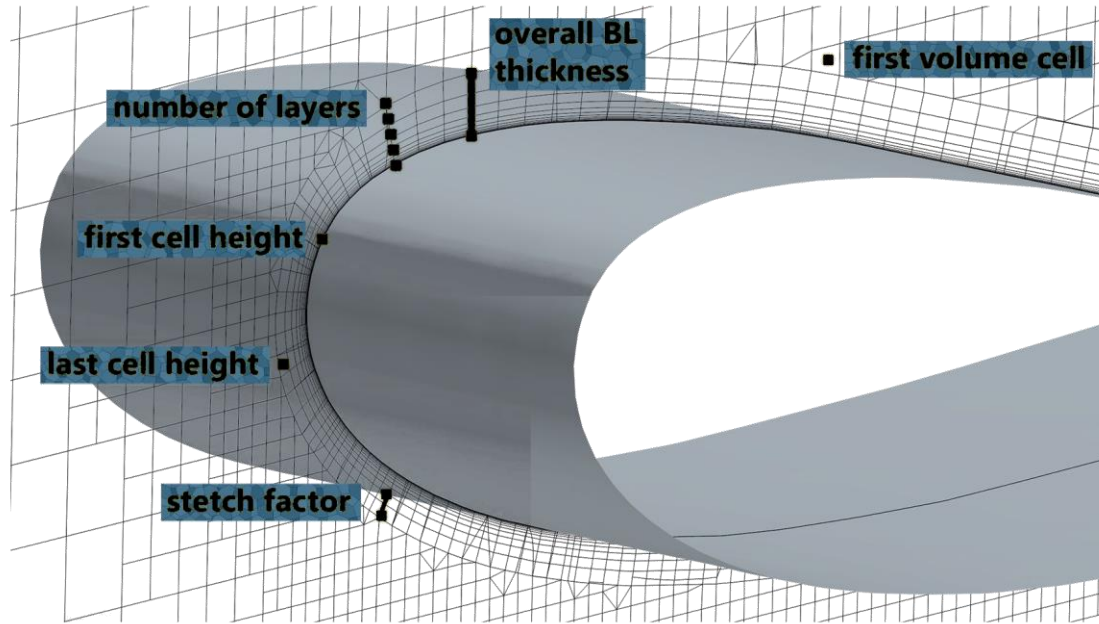
Blended Velocity

Velocity in log-layer
(turbulent part of BL)

Normalized Wall Distance

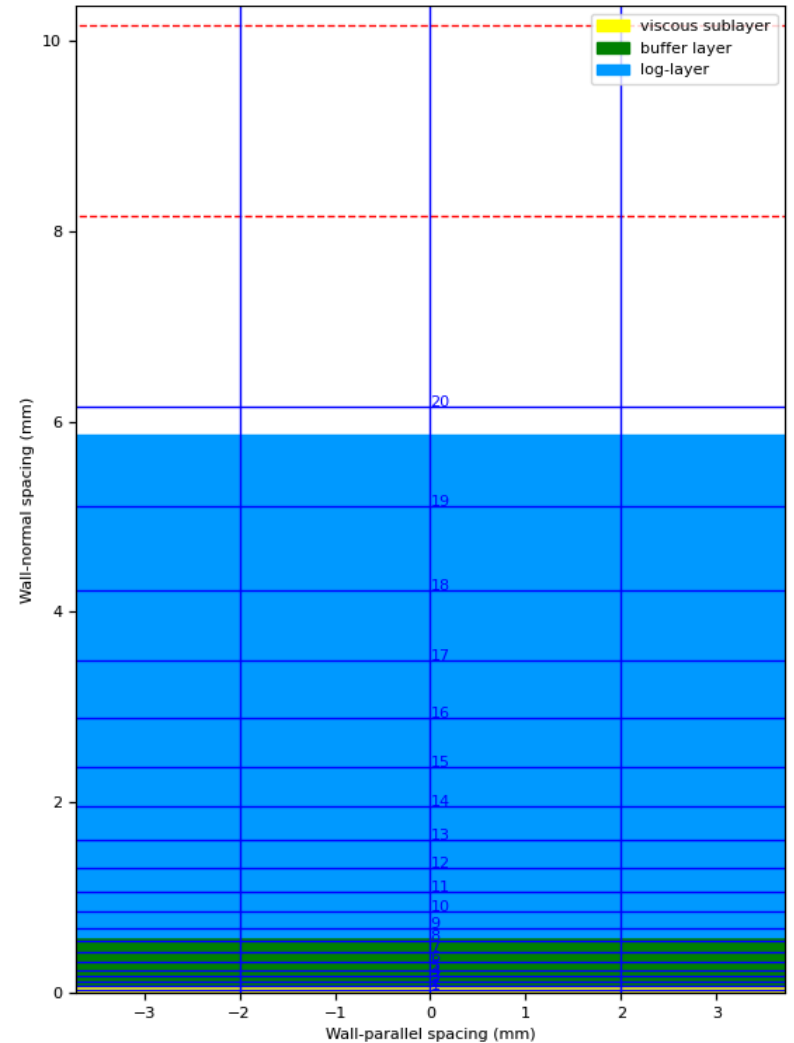
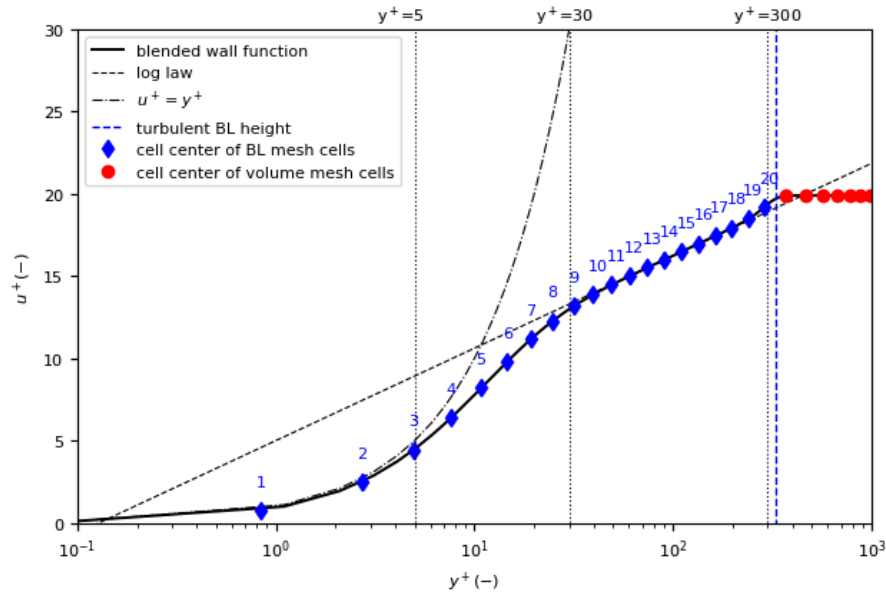
Meshing

Near Wall Meshing: Boundary Layer Mesh



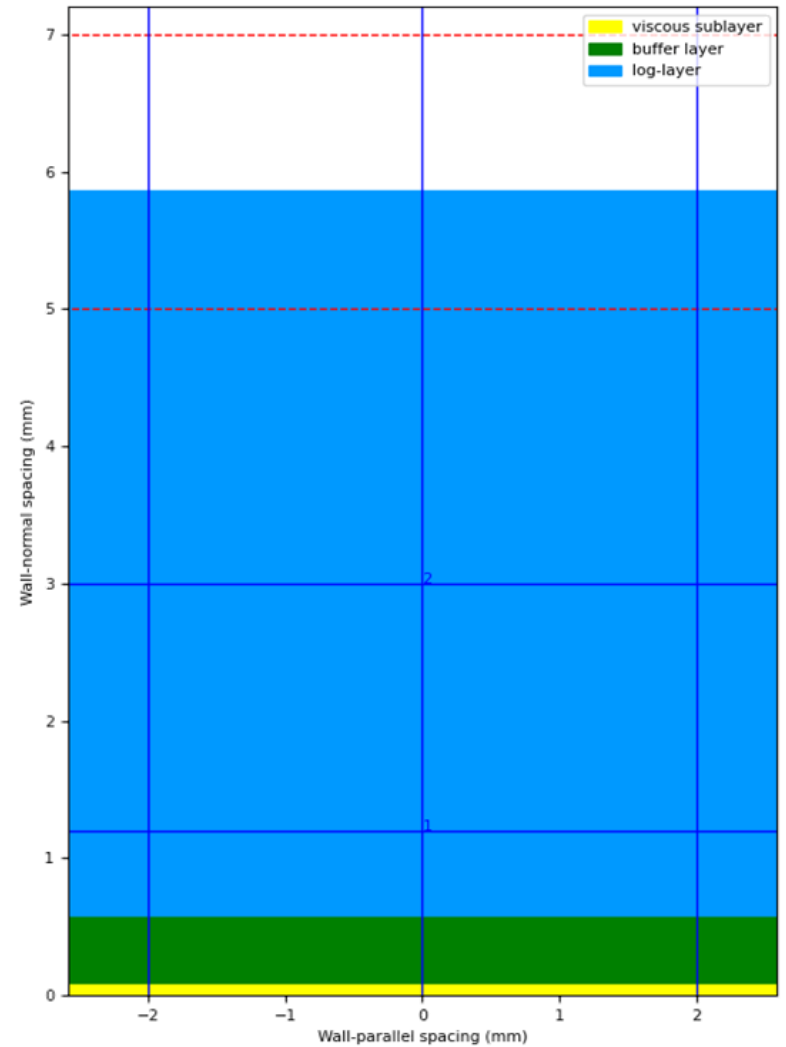
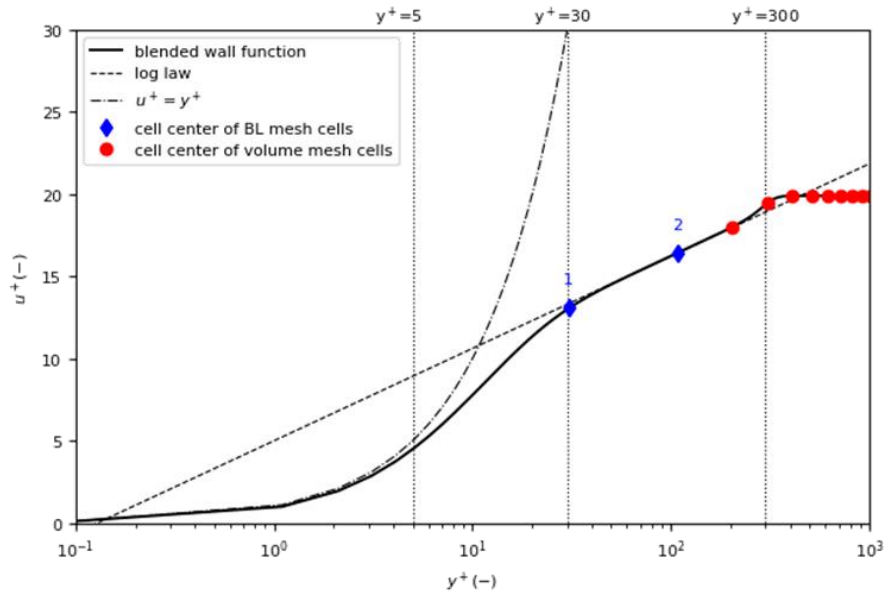
Meshing

Low y^+ Mesh



Meshing

High y^+ Mesh



Limitations and Errors

Numerical

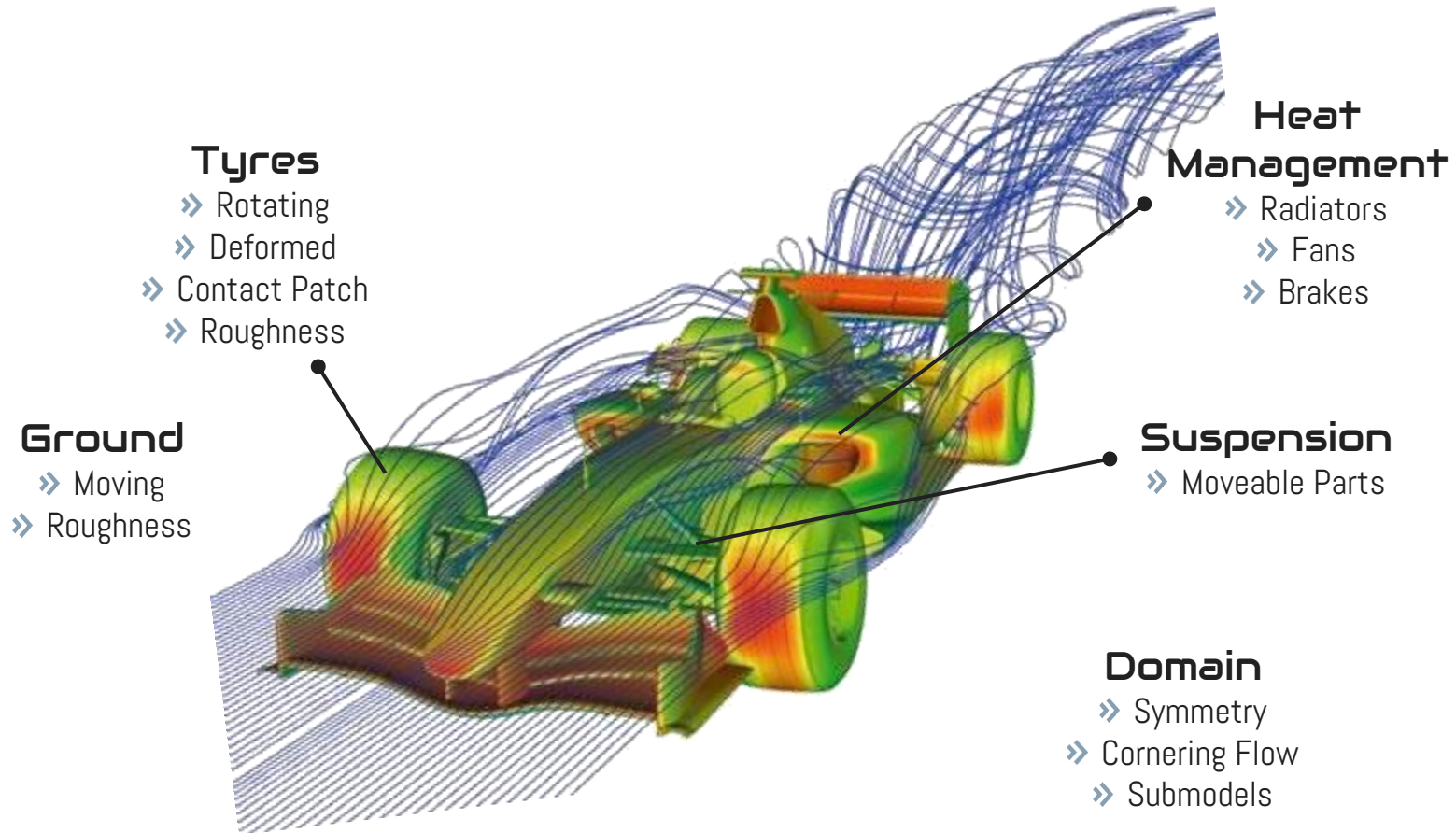
- Turbulence modelling
 - bluff body flows and highly separated flow
 - separation and reattachment points
 - high streamline curvature
 - transition modelling
- Discretization errors
 - mesh resolution
 - mesh quality
- Convergence
- Simplified Physics

Boundary Conditions and Geometry

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

2

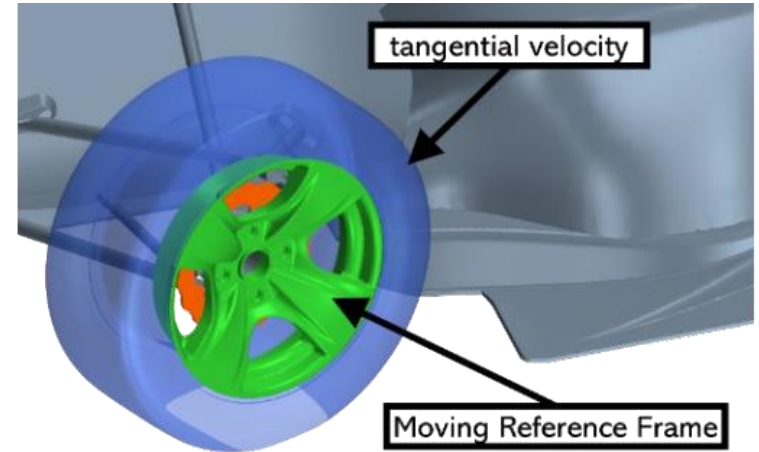
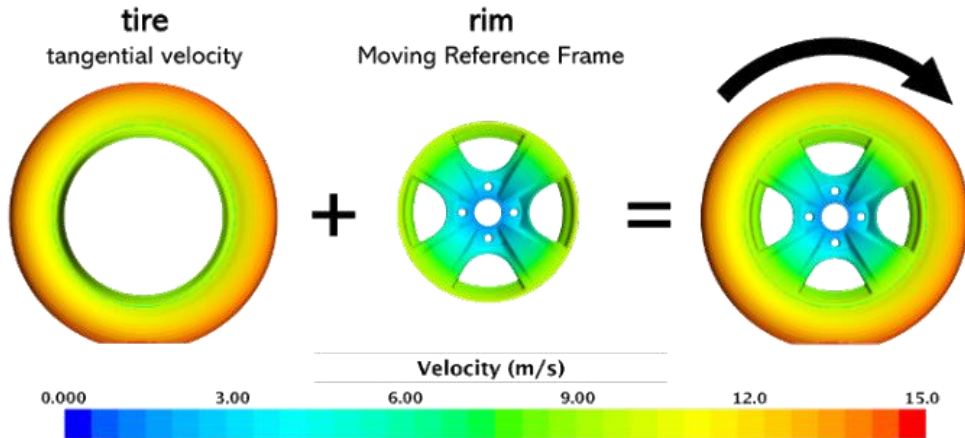
Race Car CFD



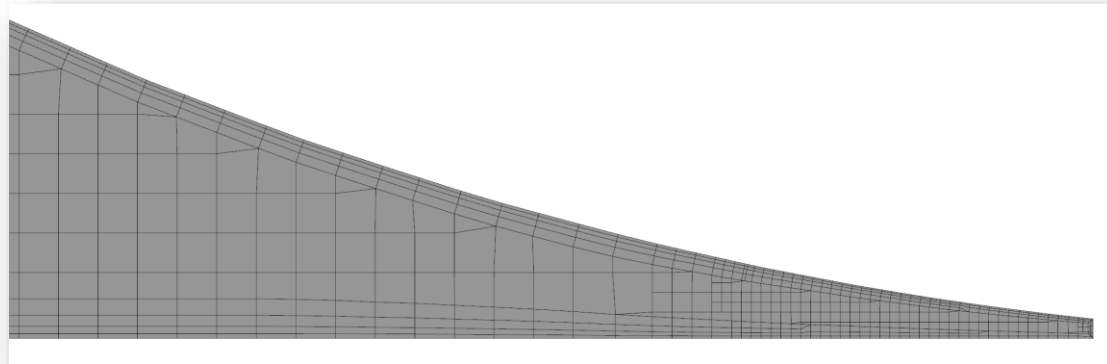
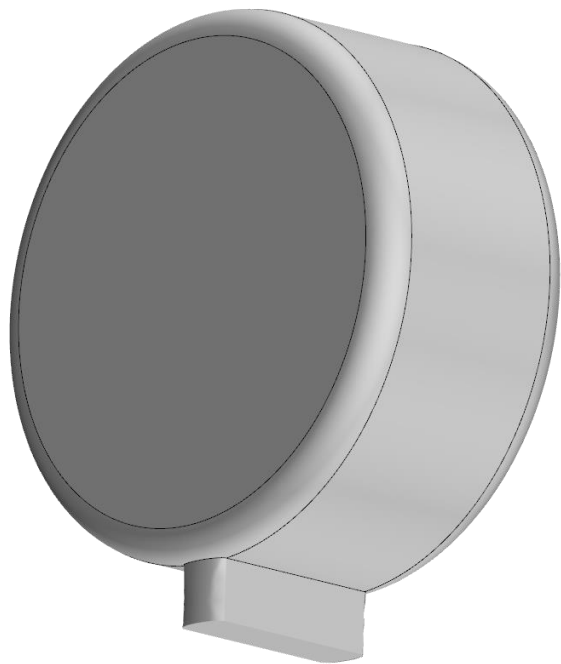
Tyres

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

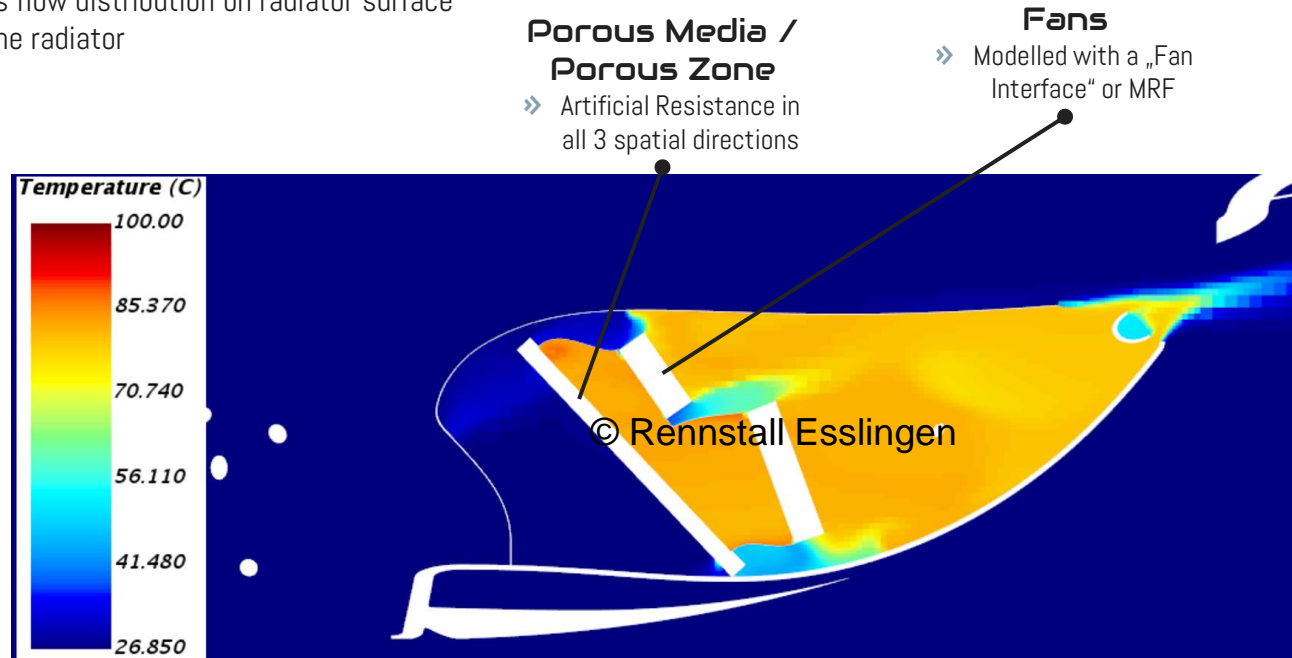


Tyres



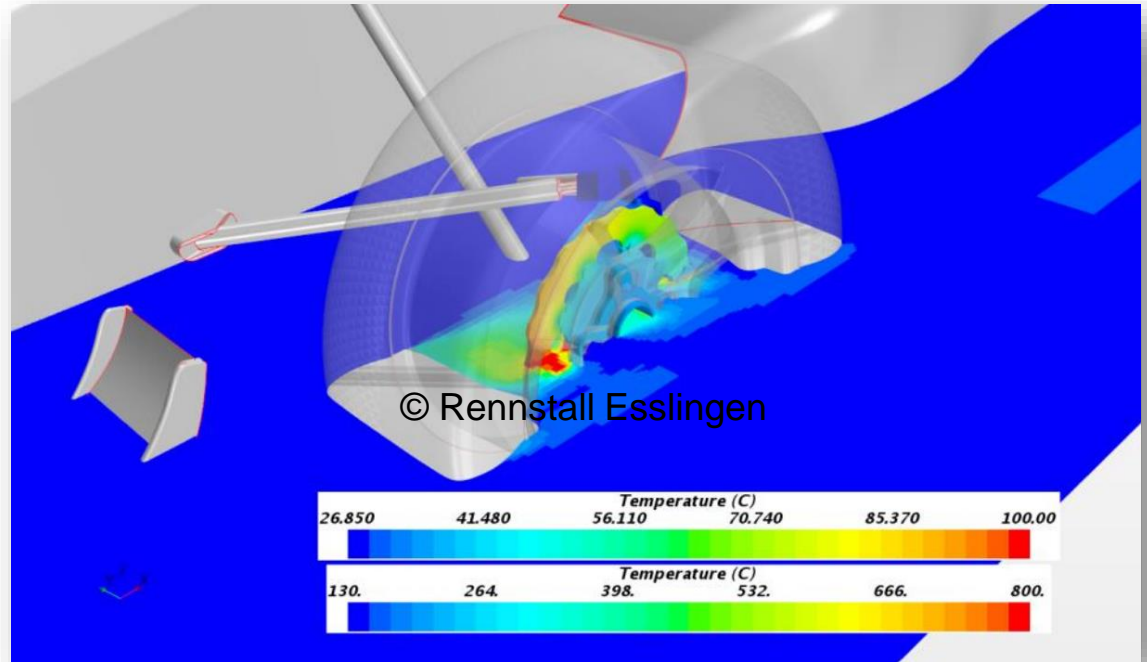
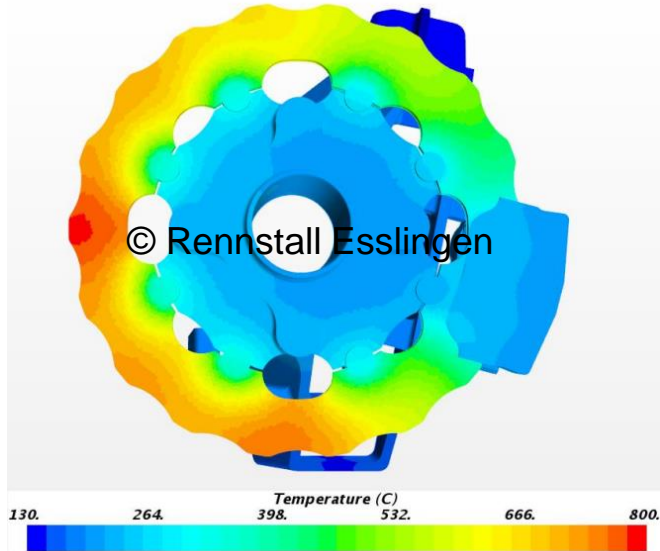
Heat Management

- 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



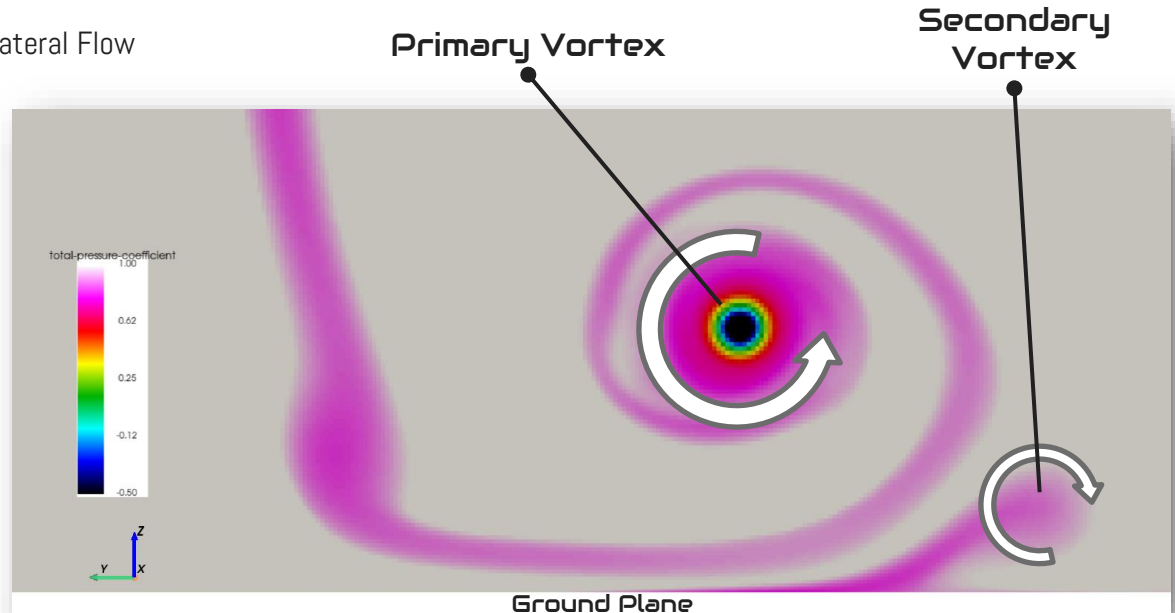
Heat Management

- Brake Cooling



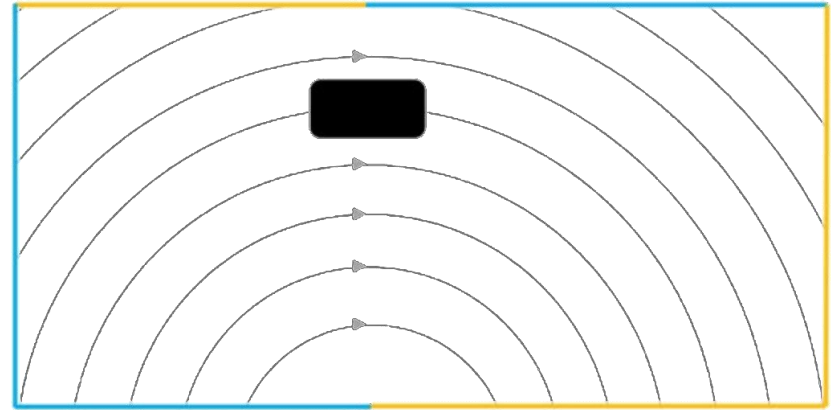
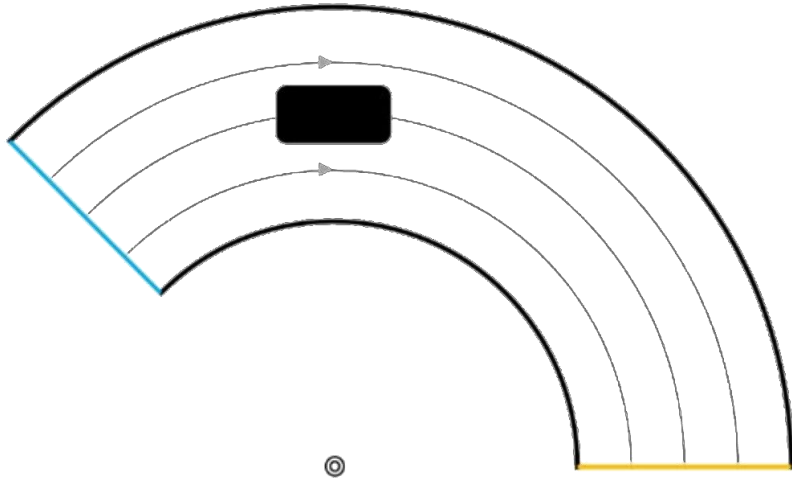
Ground

- 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”

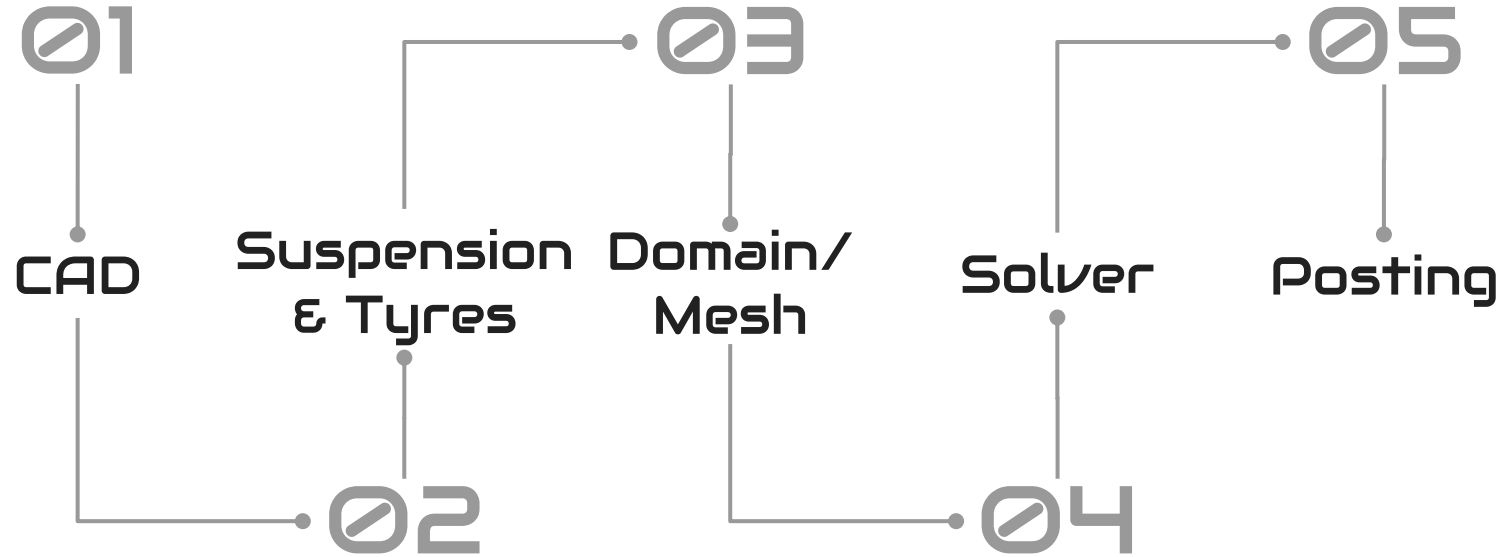


Inlet
Outlet
Walls



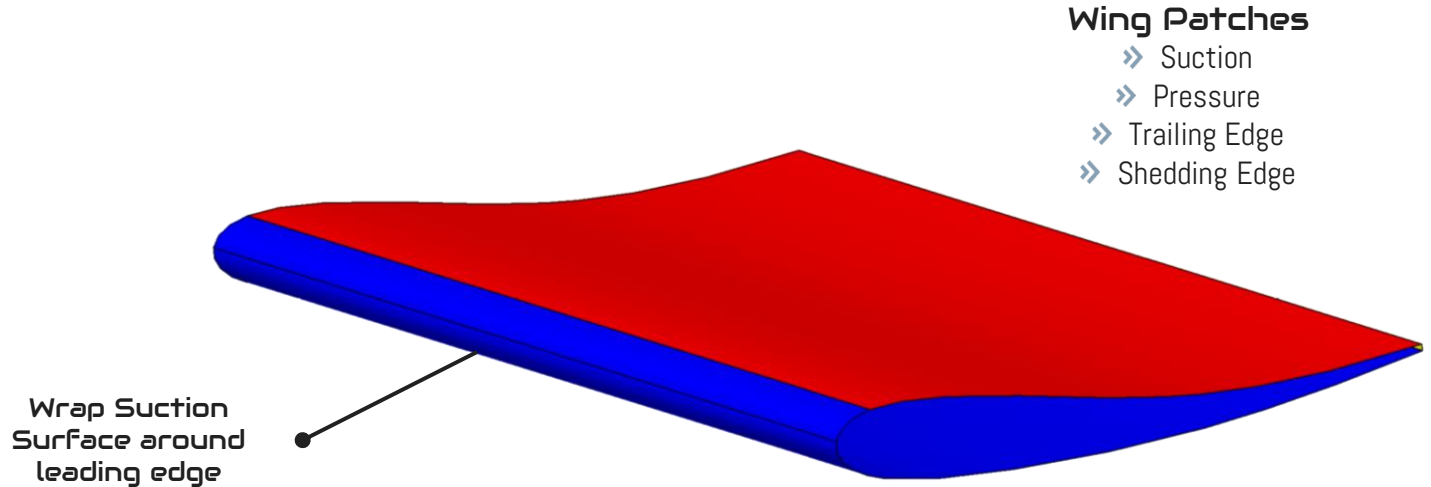
A Workflow For Formula Student

Process Overview



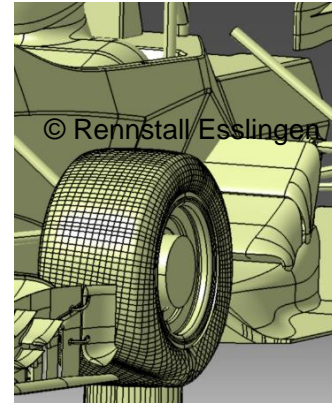
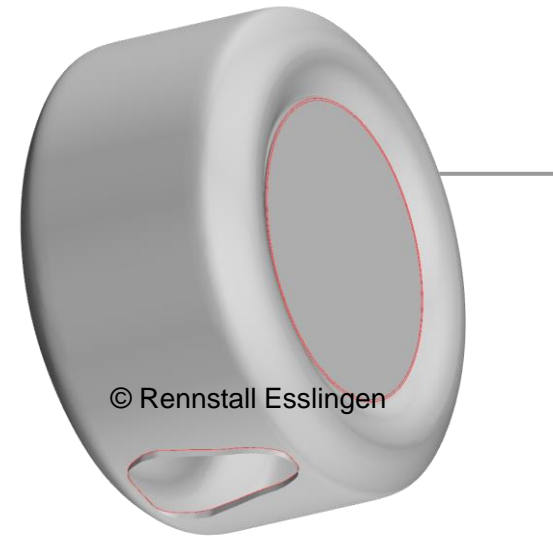
01 CAD

- 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

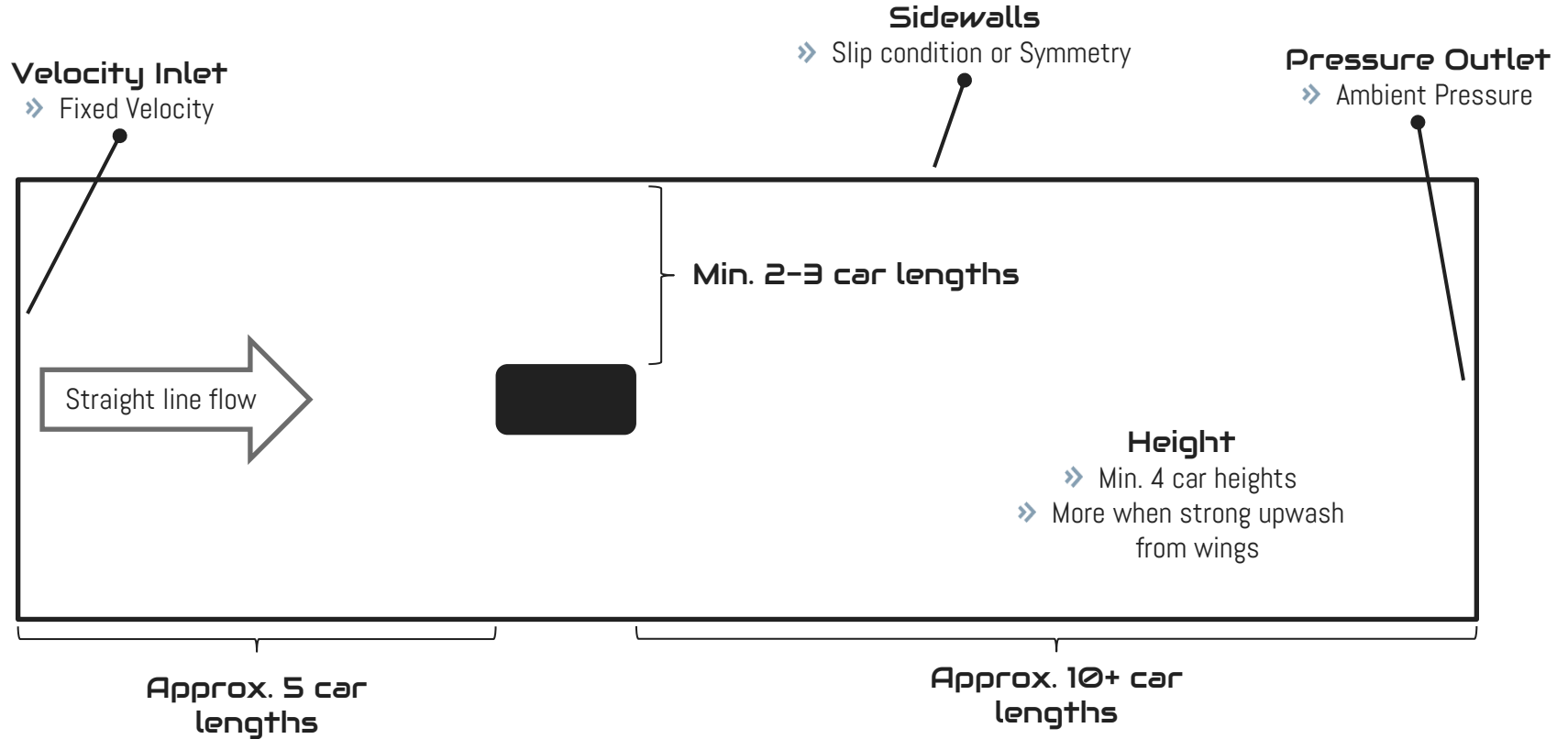


02 Suspension & Tyres

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



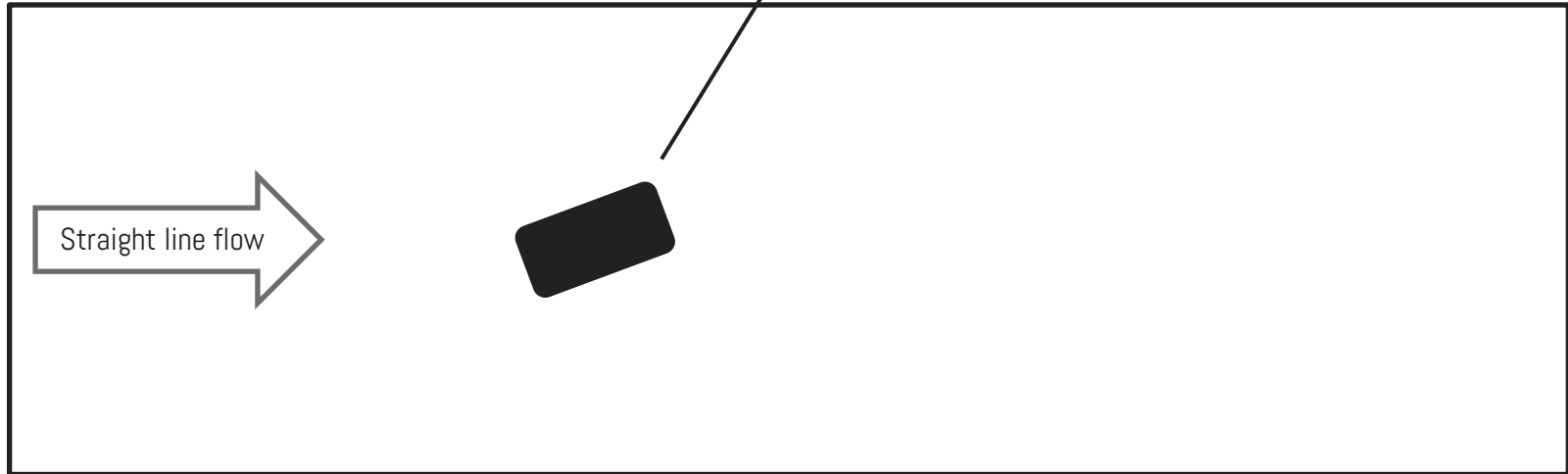
03.1 Domain



03.1 Domain

Yaw Simulations

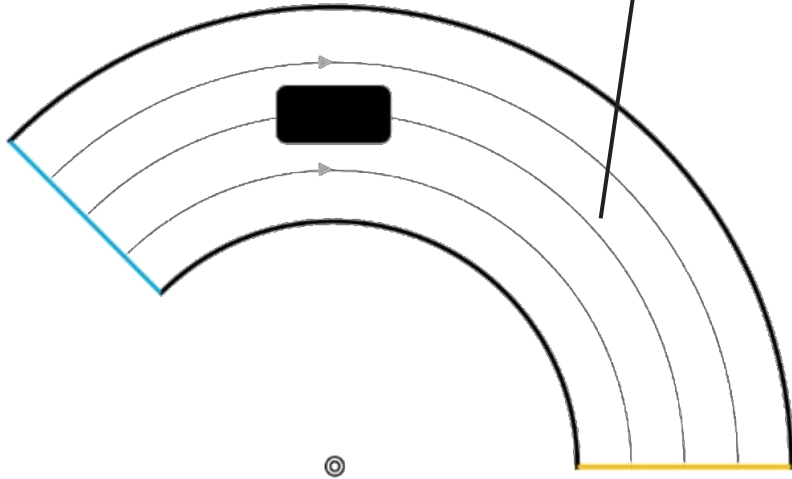
- » Rotate Car for Yaw Simulations
- » This is not a cornering simulation !



03.1 Domain

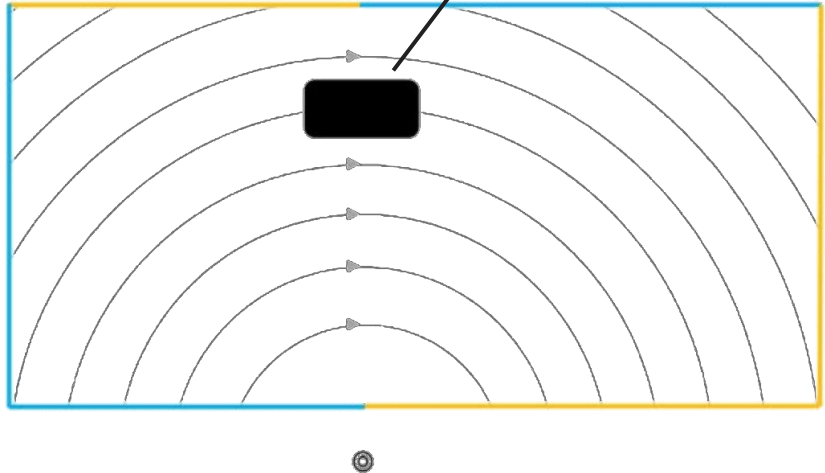
Use a Moving Reference Frame to model the curved flow

- » Change Inlet Boundary Condition to velocity = 0



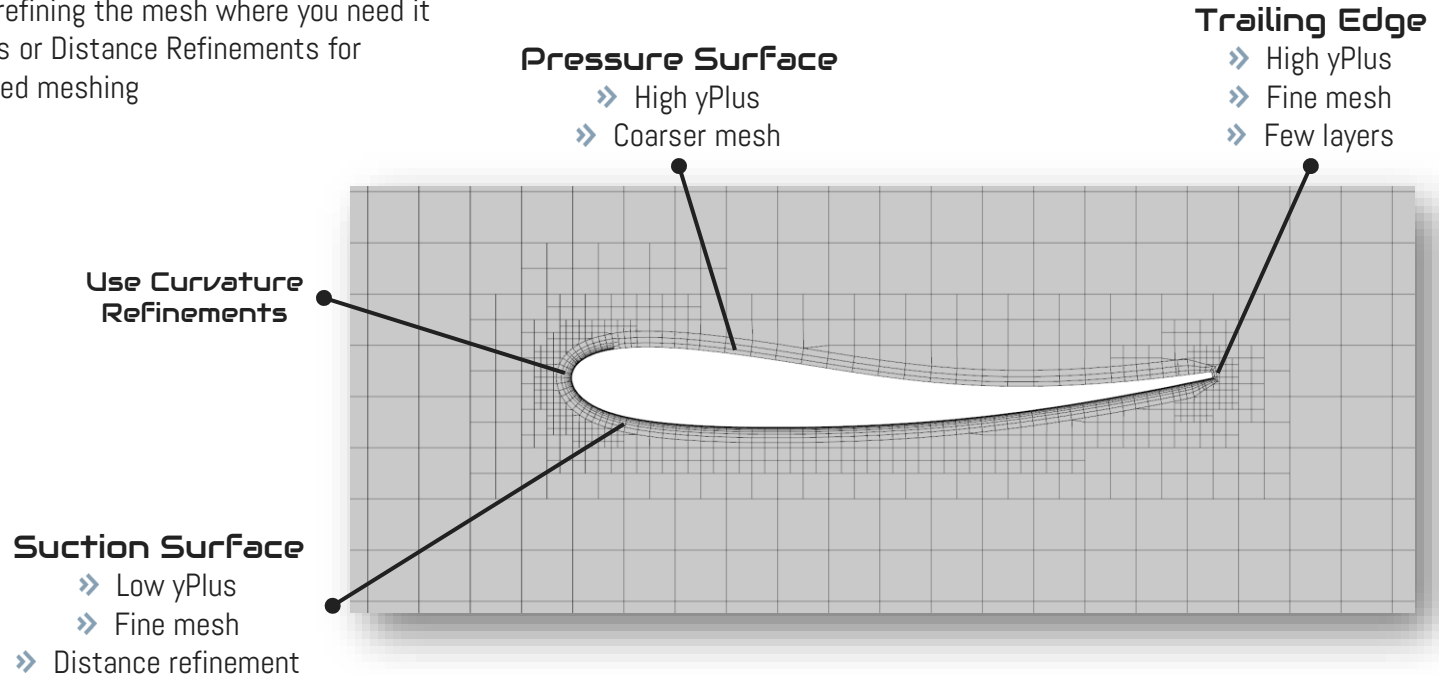
Rotate Car to account for Slip Angle

- » 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



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

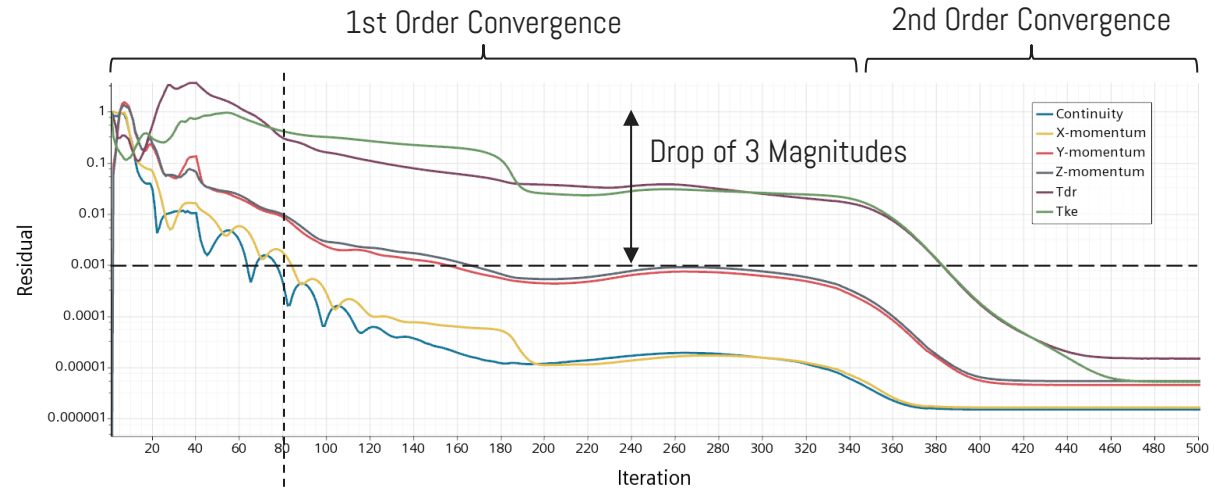
Solutions for limited computational resources

- 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!

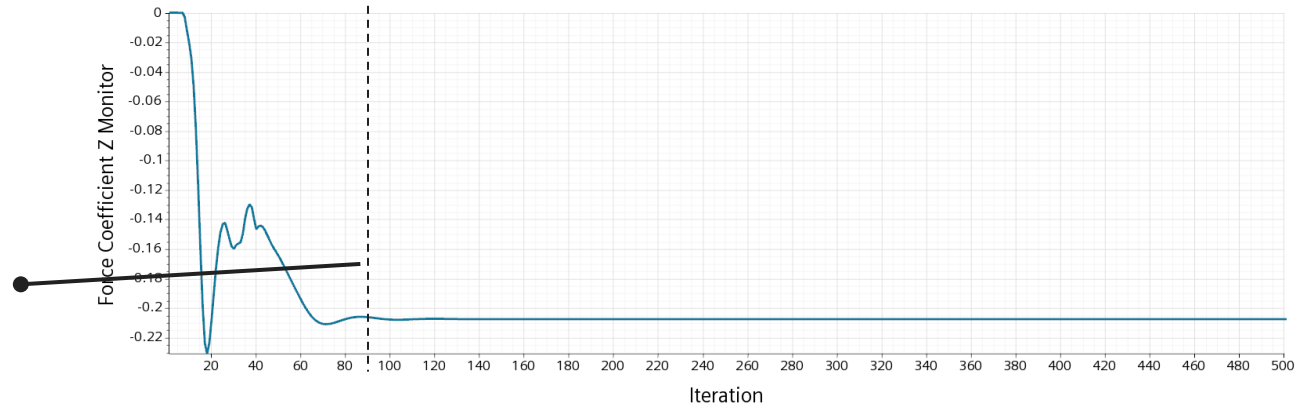
04 Solver

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

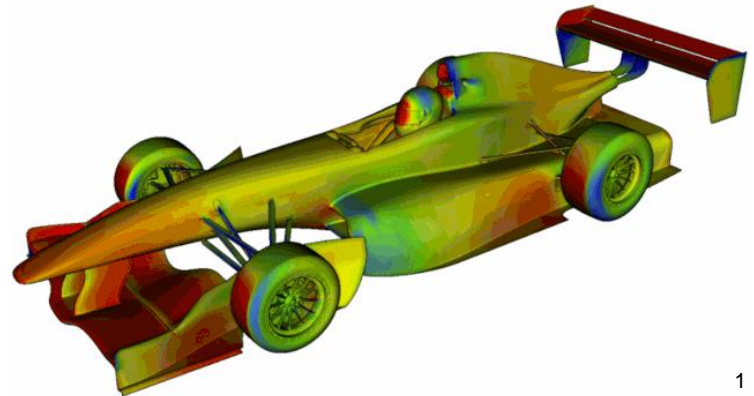
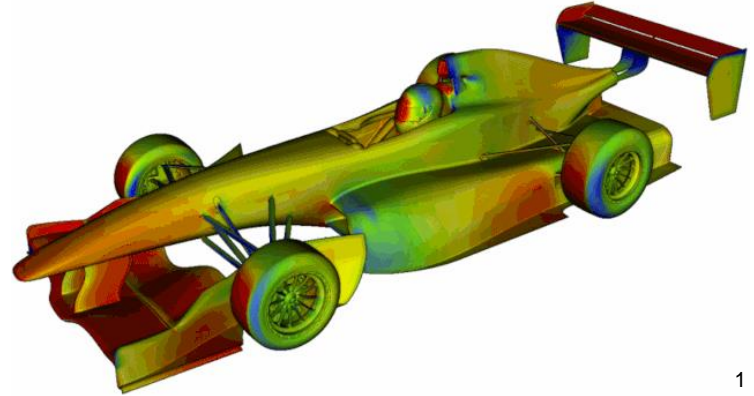


05 Postprocessing

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!

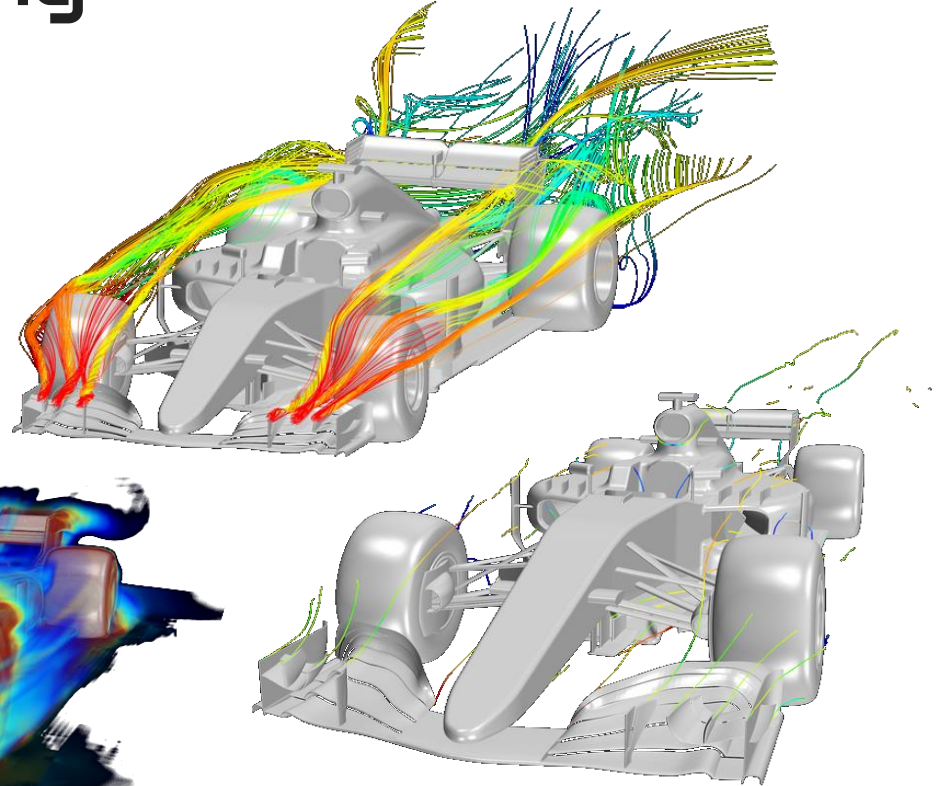
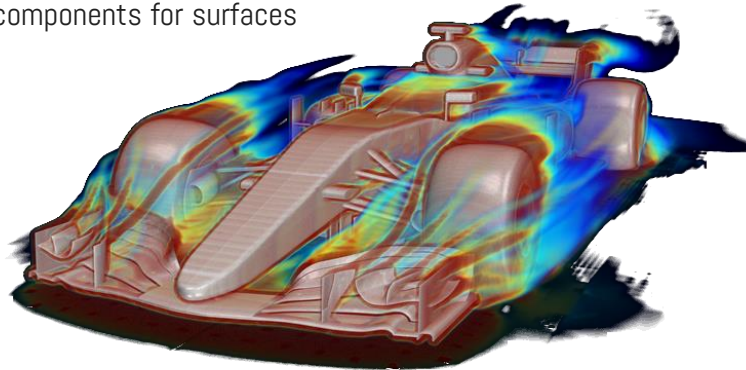


05 Postprocessing

Flow Field Visualization

1. Understand the flow field

- Streamlines
- Vortex Cores
- Iso-Surfaces
 - Total Pressure
 - Vortex Criterion: Helicity/ Q or λ^2
- Line Integral Convolution (LIC)
 - Velocity components for cutting planes
 - Skin Friction components for surfaces

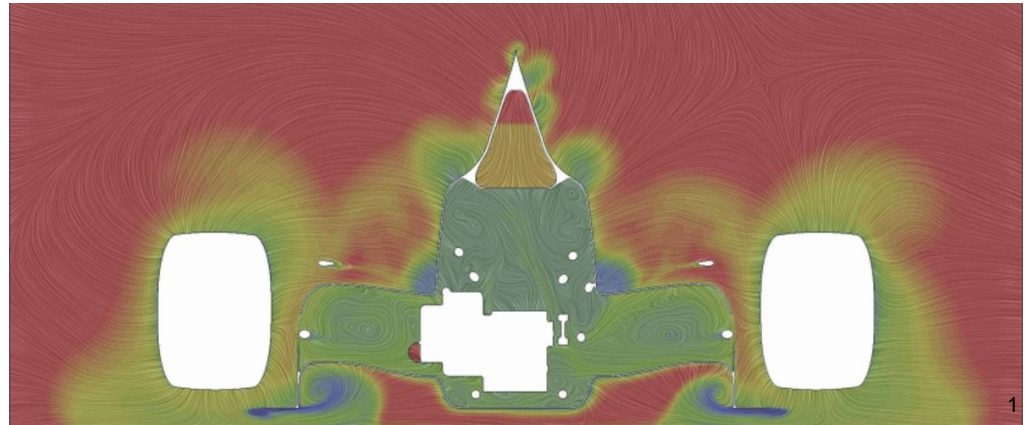
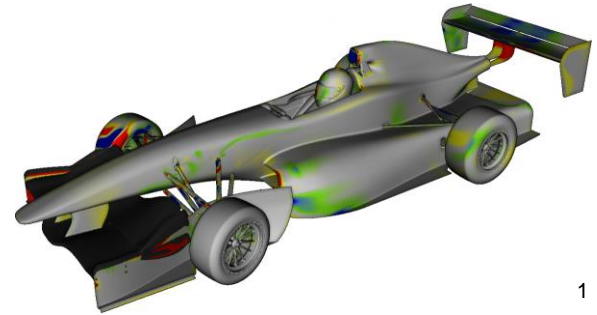
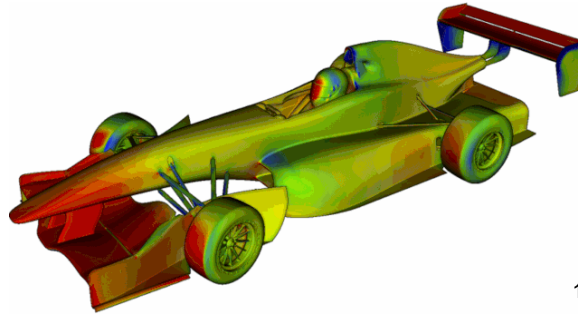


05 Postprocessing

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

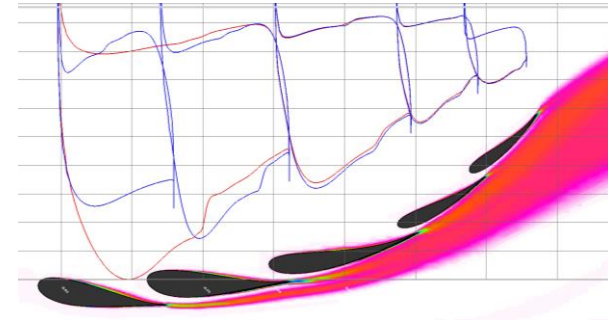
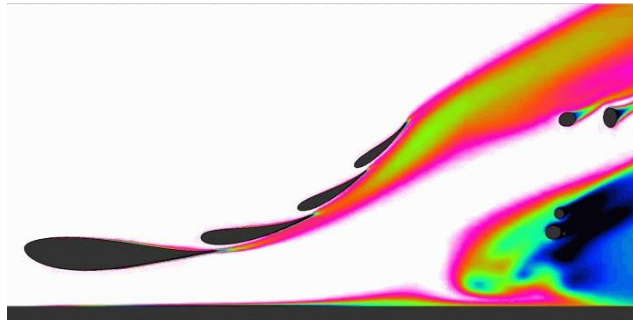


05 Postprocessing

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



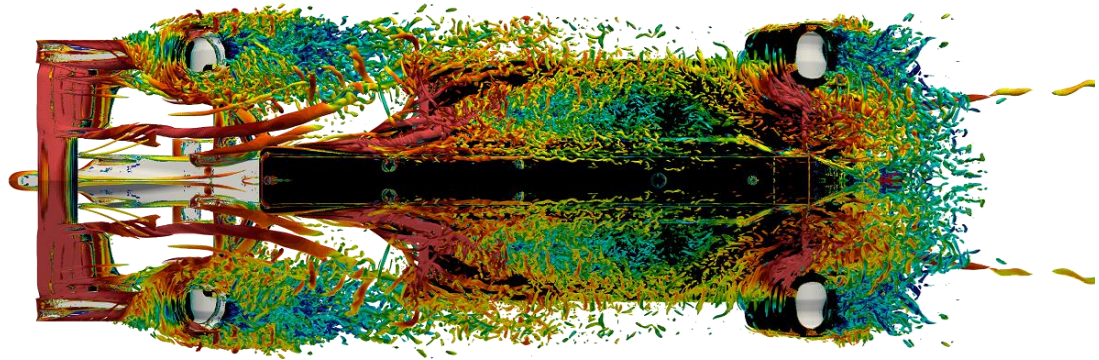
4

Validation and Correlation

Data Collection

Where to get your data to correlate to from?

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



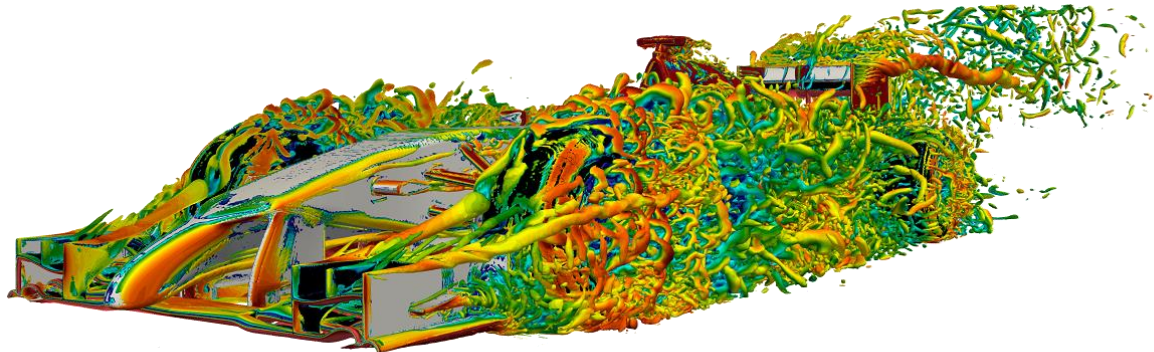
Data Collection from CFD

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

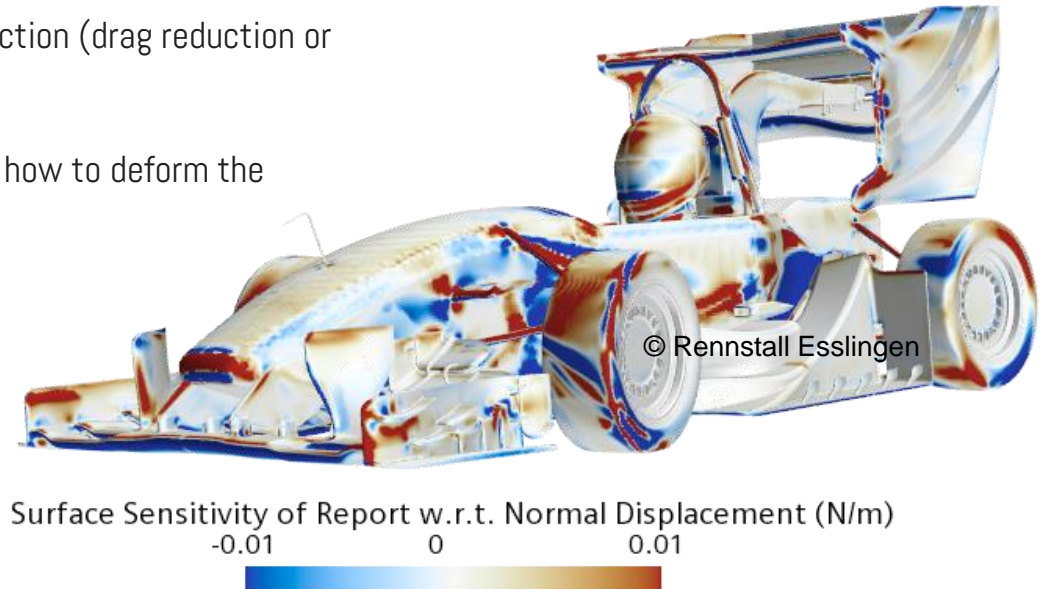
- » Check Geometry
- » Mesh sensitivity
- » Check Curvature Correction is activated and check parameters
- » Adjust turbulence model parameters

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**

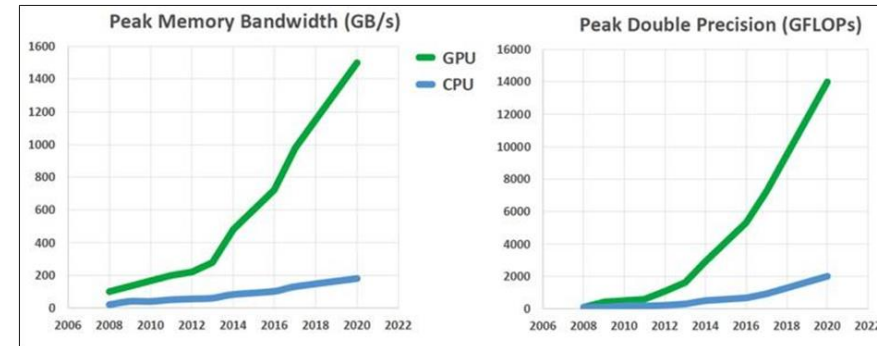
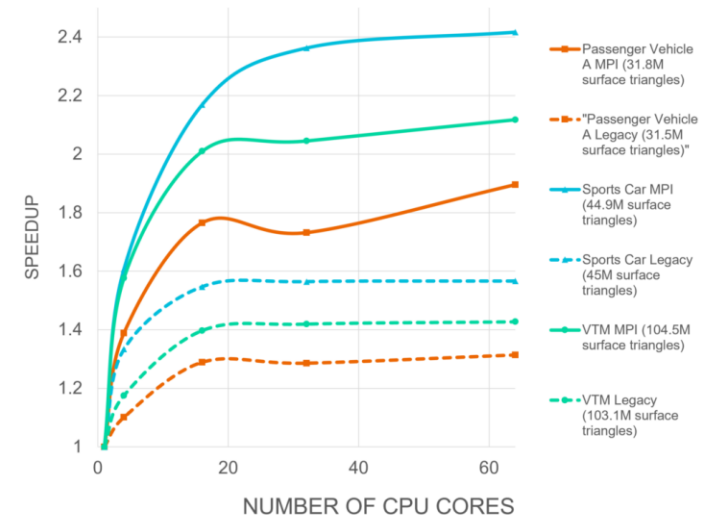


GPU Based Solving

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

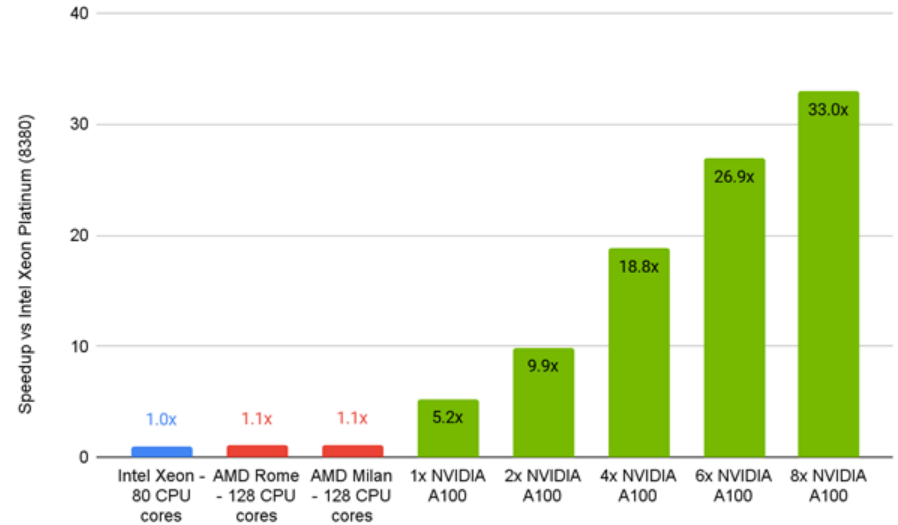


GPU Based Solving

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

