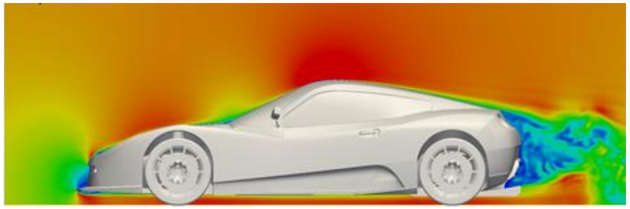
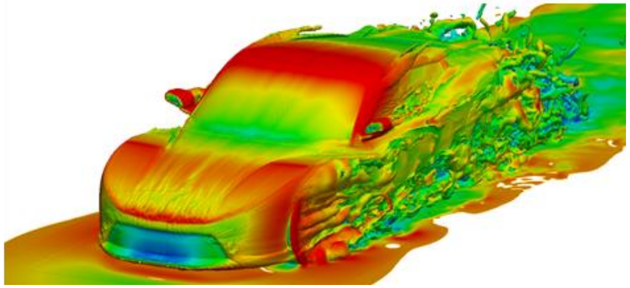
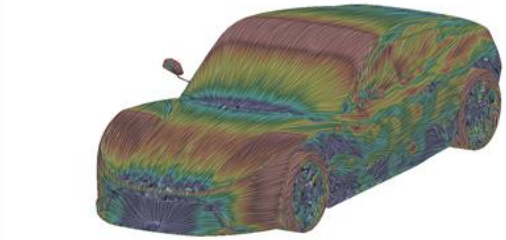
Validation of an Automated Process for DES of External Vehicle Aerodynamics







DANSIS

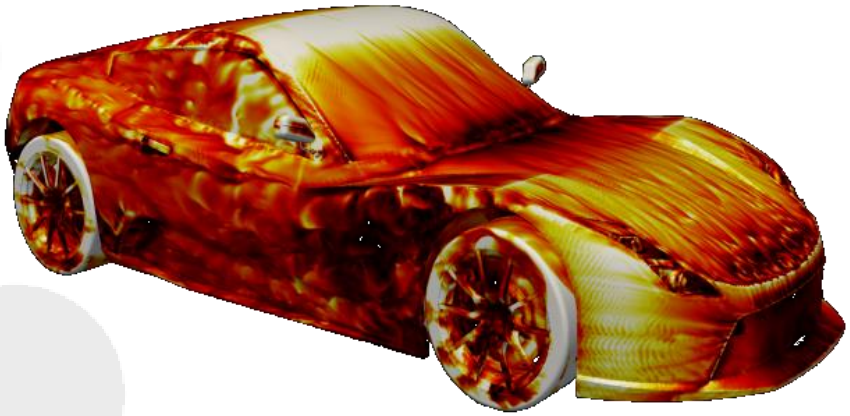
Automotive Fluid Dynamics

25 March 2015

Eugene De Villiers

Contents

- Development Driving Force
- CFD Methodology
  - Pre-processing
  - Flow Solution
  - Post-Processing
- Example: Fiat Bravo
- Validation
- Further Developments



Development Driving Force

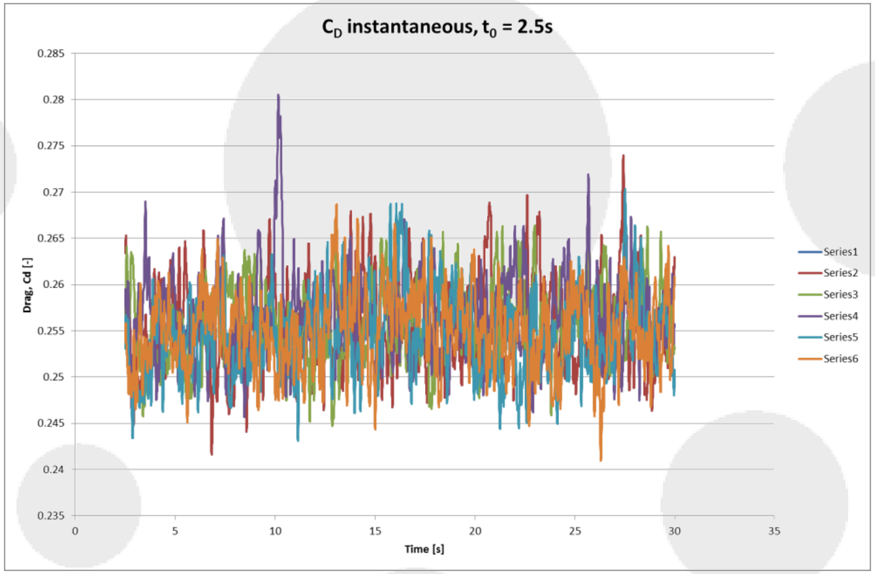
- Why is there a need for another CFD tool?
- Regulatory pressure
  - US: CAFE 40-60% reduction by 2025 (target 101g/km)
  - EU: 95g/km fleet average by 2021 (currently 130g/km)
- Evolving technology
  - Electric PT + KERS: aerodynamic losses ~40% → 62%
- Shortening design cycle
  - Programmatic codification of simulation process
  - Improved interface between styling/engineering/management
  - Integrated Optimization

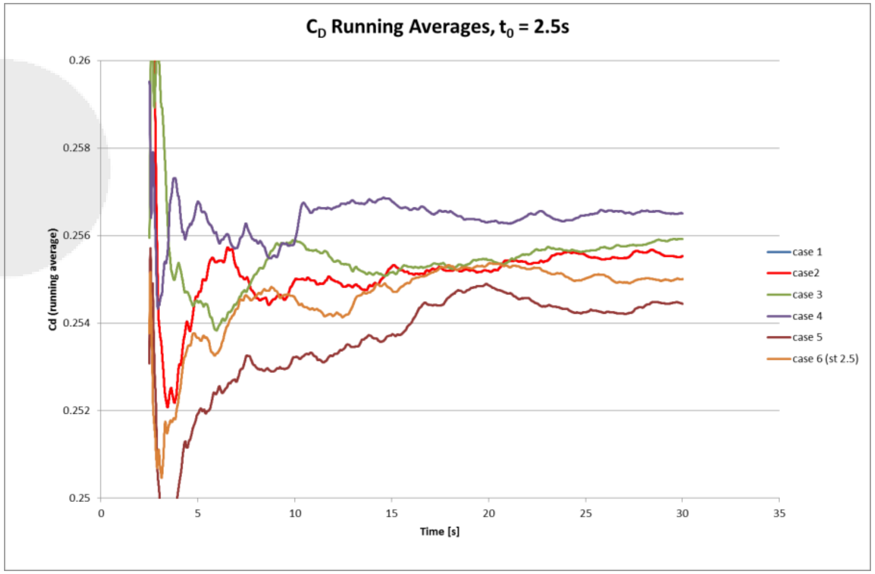
Development Driving Force

- Requirement → modeling/simulation method that is:
  - Validated against experimental results
  - Reproducible and stable (for all vehicle shapes)
  - Computationally efficient
  - Accurate physically meaningful results
- RANS based methods are efficient, but not consistent
  - Accuracy for RANS vs. experiment is highly case dependent
  - ΔC_d ranges from -10 % → +25% have been experienced
  - More pronounced in bluff vehicles (hatchback, SUV, truck, etc.)
  - Does not produce consistently correct trends

Development Driving Force

- DES (Detached Eddy Simulation)
  - Combines best aspects of RANS and LES
  - Can produce consistently accurate results
  - Computationally expensive
    - Highly oscillatory – requires long integration times to reduce uncertainty





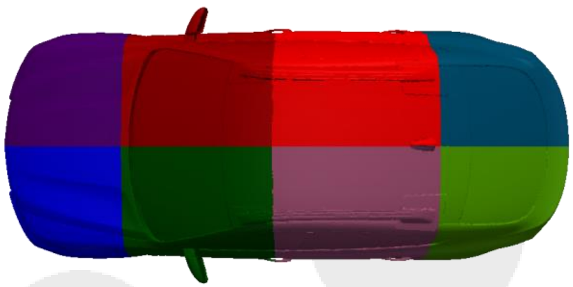
Development Driving Force

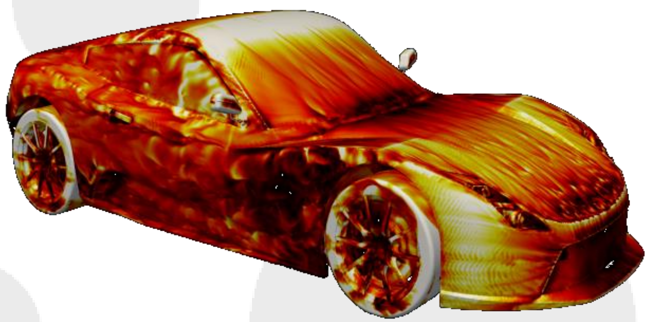
- Strategy
  - Given: simulation costs are high
  - Address other bottle-necks (Mesh, Setup, Post)
  - Optimise settings for acceptable accuracy/speed/stability
  - Automated, well validated workflow and best practices

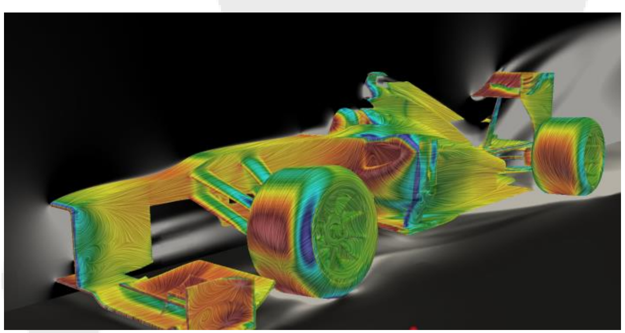
Mesh Generation → **Flow Modeling** → **Post-Processing**

Automated workflow based on best practices

CFD Methodology | Overview







Pre-Processing
- Wizard-like GUI
- Hex-dominant Cartesian mesher
- Fully parallel

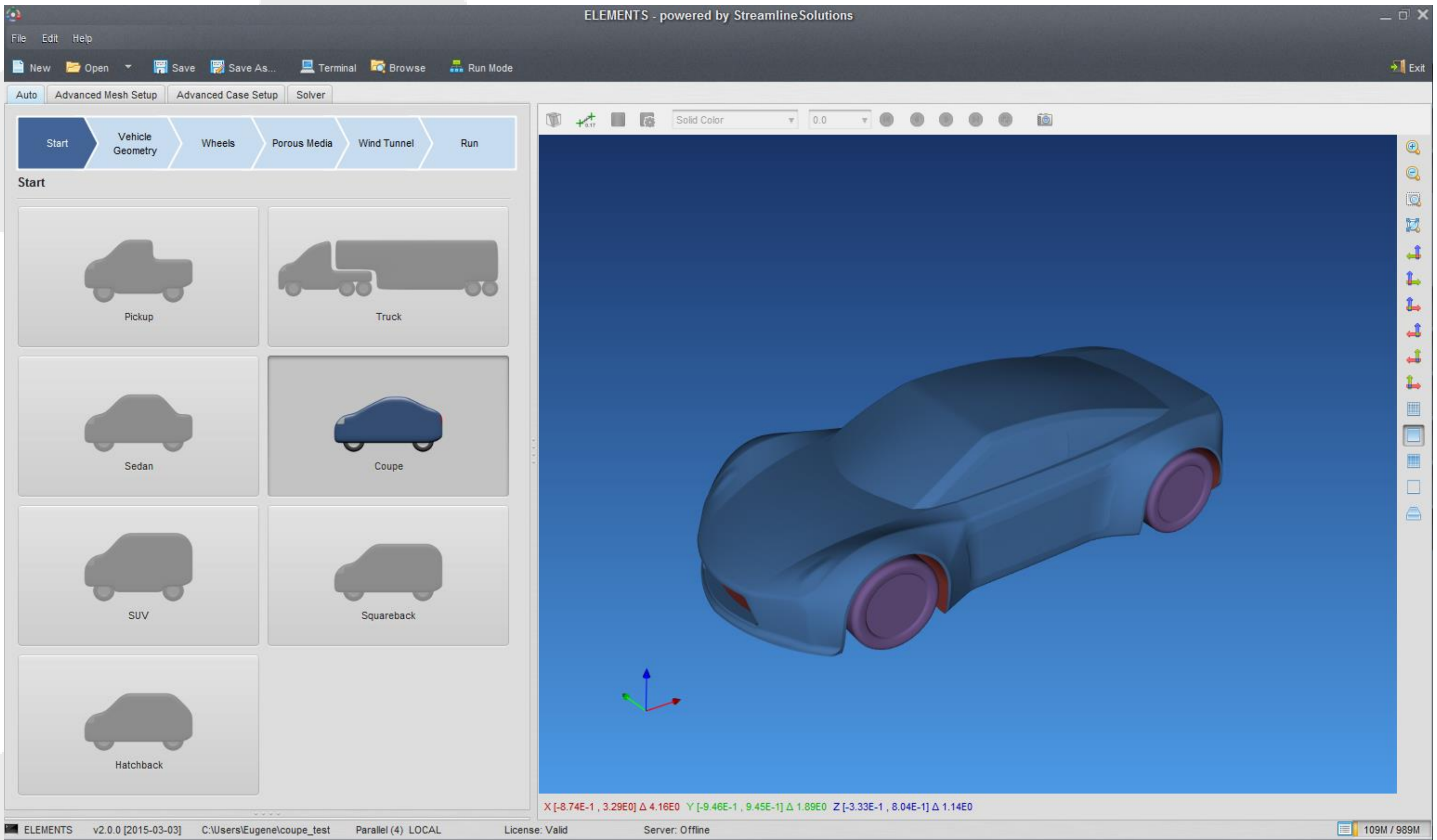
Flow Modeling
- Based on open-source technology
- Accurate, robust and HPC ready

Post-Processing
- Parallel data analysis tools
- Automatic report generation
- Customizable scripts

engys

CFD Methodology | Pre-Processing

- ELEMENTS Vertical Application
  - Process driven interface for automotive simulation



CFD Methodology | Pre-processing

User Input

- CAD → Surface Mesh
- Select vehicle type
- Load geometry
- Categorize Parts
- Review wheel settings
- Enter porous coefficients
- Select wind tunnel
- Execute

Automatic

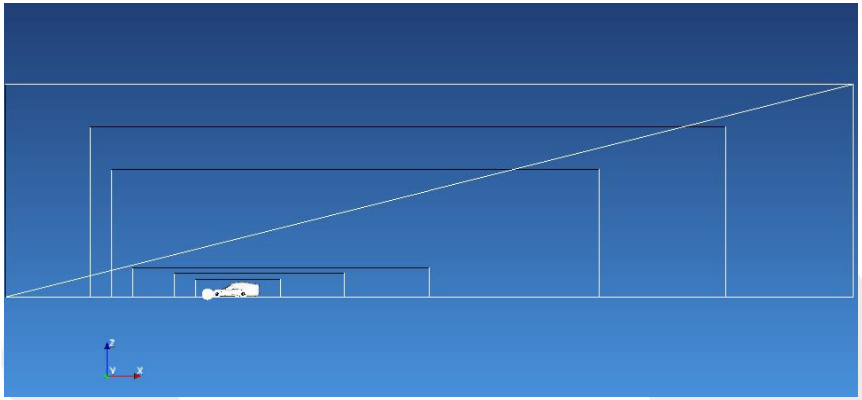
-
- Load best practices
- Apply transforms
- Create meshing input
- Detect C.o.R.
- Calculate local
- Configure flow BCs
- Mesh, setup, solve, post

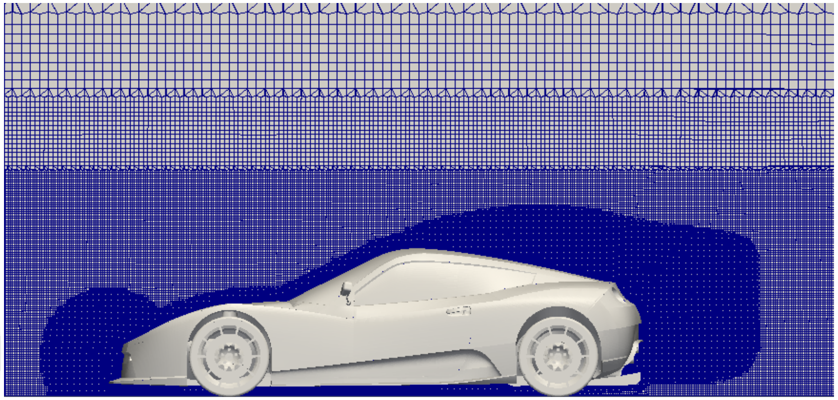
Workflow

- Minimal cleanup prior to loading
- Vehicle type and dimensions influence parameters
- Geometry scaled appropriately
- User assigns predefined categories that dictate treatment
- Wheel orientation and location detected and verified
- Porous regions are defined and verified
- Wind tunnel information stated and BCs set automatically
- Combined single run from mesh → post-processing

engys

CFD Methodology | Mesh Generation



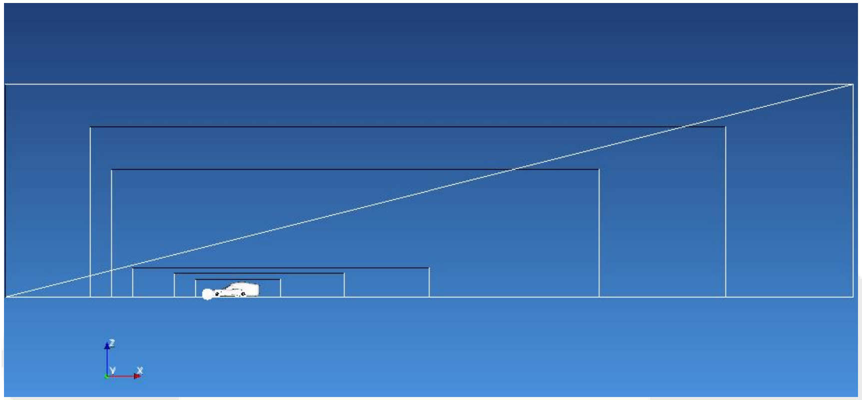


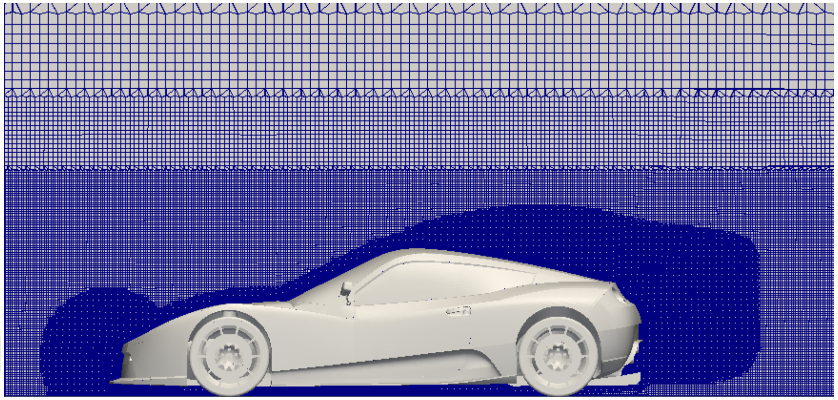
Modified snappyHexMesh

- Heavily optimised for improved speed and robustness
- Minimal CAD preparation required
- Improved cell quality
  - optimisation, cell splitting
  - collapse of degenerate elements
- Automated feature capturing
- Improved solver robustness

With kind permission of: Helsinki Metropolia University of Applied Sciences www.metropolia.fi
RaceAbout Association www.raceabout.fi

CFD Methodology | Mesh Generation



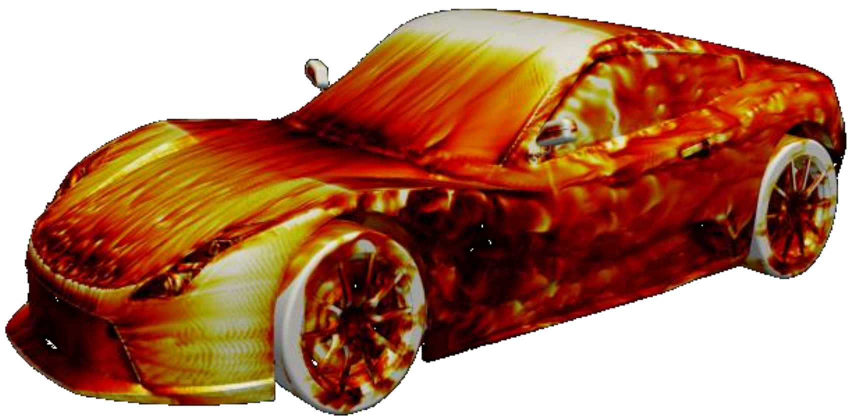


Meshing based on global and vehicle specific best practices

- Wind-tunnel size and positioning
- Tailored mesh refinement
  - Surface- and proximity-based cell-spacing
  - Volumetric refinement boxes and buffer layer refinement
  - Stagnation and wake zone refinement tailored for each vehicle type
  - Additional refinement levels for capturing separation (e.g. hatchback D-pillar)
  - Contact patch refinement via cylindrical plinth
- Optimised number and placing of near-wall layers on vehicle surface

CFD Methodology | Flow Solution

Transient DES formulation



- Improved PISO algorithm
- 2nd order pressure boundaries
- Spalart-Allmaras DDES model
- Wall-function-aware near-wall production term
- Model consistent wall functions for Spalart-Allmaras (Knop)
- Corrected shield function, Ψ (GIS), reduces build-up of energy at smallest (grid) scales
- Improved SGS length scale formulation supports WMLES

engys

CFD Methodology | Flow Solution

Initialisation

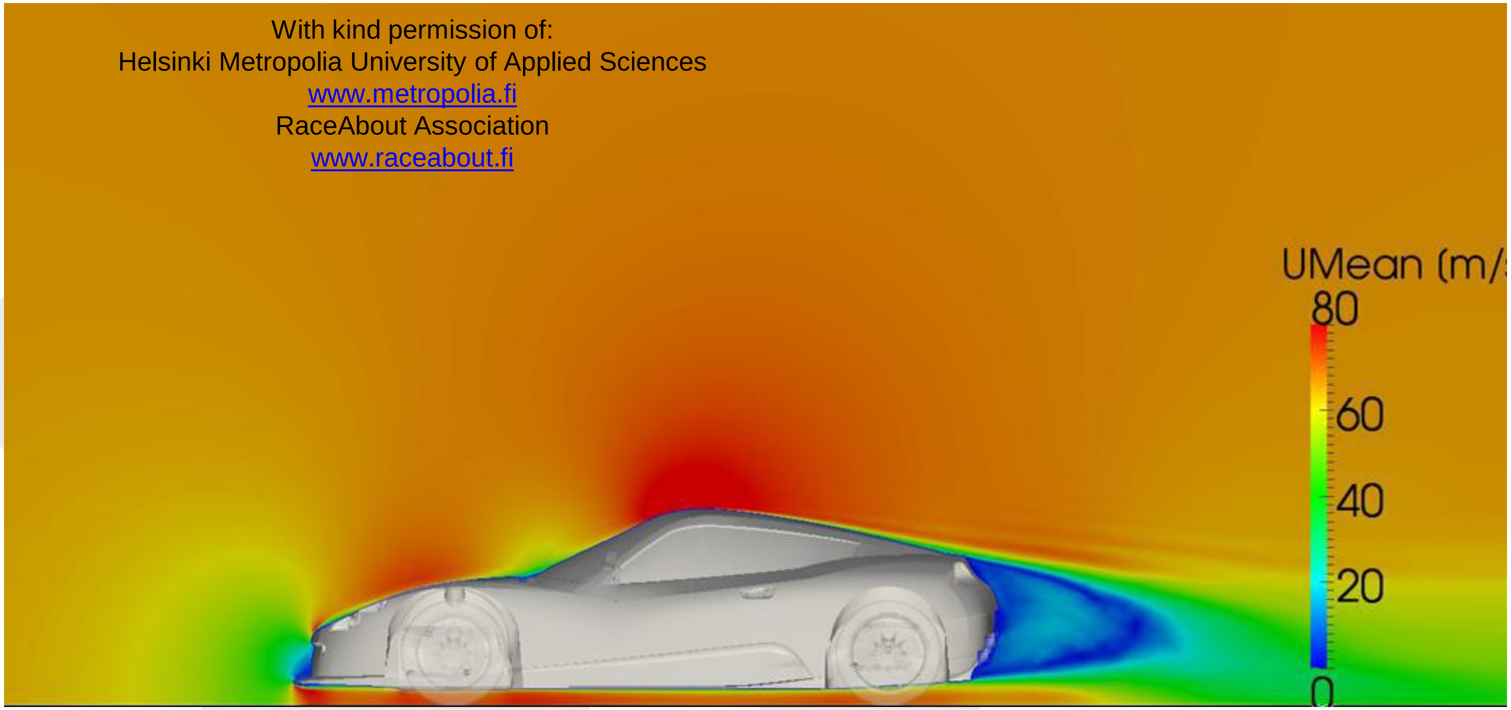
- Large time step (5e-3), more diffusive 2nd order discretisation
- Adaptive non-orthogonal correction during initial steps
- Flushing of initialisation transients
- Allows development of wake and large scale structures
- Vehicle type and size dependent integration period
  - Bluff/Larger vehicles require longer initialisation
- More efficient and robust than coarse mesh + mapping

CFD Methodology | Flow Solution

High fidelity calculation

- Optimised time step size (vehicle dependent, ~ 2e-4s)
  - Biggest factor determining compromise between accuracy and speed
- Integration time
  - Larger vehicles require longer averaging to reduce uncertainty range
  - Scaled with free stream velocity
- Robust blended 2nd order advection scheme
  - Near-wall RANS region: fixed blend of CD & LUD
  - LES zone: adaptive blending of filtered CD scheme and LUD based on CFL
  - Increased numerical diffusion at refinement interfaces, far field

CFD Methodology | Post-processing



Automated post-processing:

- Customisable pdf format report
- Contour plots of U, p, tauw, Unw fields
- Isosurfaces of $P_{tot,}$ Averaged C_D and C_L, C_p

Validation Process | Overview

- In partnership with A.R.C.
  - Wind tunnel and model shop available
- Over 100+ test cases:
  - Different vehicle platforms
  - Different brands
  - Different wind tunnels (full size and scale)
  - Multiple parametric changes
- Mesh sizes from ~29M to 101M Cells
- Execution times ranged from 18h to 49h
- Meshing strategy, modelling parameters and solver settings all tuned to define “Best Simulation Practices” for each vehicle shape

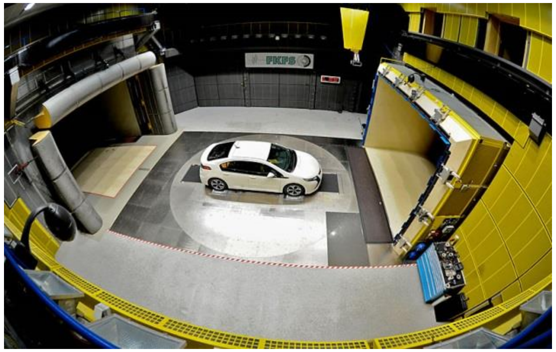
Validation Process | Vehicle Types

- Multiple vehicle types:
  - Sedan
  - Hatchback
  - Estate
  - SUV
  - Sportscar
  - Streamliner
  - Nascar
  - Indycar
  - Light Duty Truck
  - Heavy Duty Truck



Validation Process | Parametric Changes

- Fixed Ground
- 5-belt Moving Ground
- Single Belt Moving Ground
- Yaw Angle
- Ride Height
- Test Speed
- Vehicle Modifications (e.g. Open or Closed Cooling, Spoiler on/off, Underbody Panels on/off)



Validation Process | Results Cars

Vehicle No.	Vehicle Model	Grille (open, closed, blanked)	Wind Tunnel Data		Elements			Notes (no. of cells, prism layers, run length, etc.)
					Coefficients			
			Scale	Ground Simulation	CD	CLF	CLR	
1	DRIVAER Estate	n/a	40%	Single Belt	2.40%			Scale Model, 29.5 Cells, 24 hrs, 3 body layers
2	DRIVAER Fast	n/a	40%	Single Belt	-0.41%			Scale Model, 29.7 Cells, 18.4 hrs, 3 body layers
3	DRIVAER Notch	n/a	40%	Single Belt	0.41%			Scale Model, 29 Cells, 24.3 hrs, 3 body layers
4	Sedan 1	open	100%	5 Belt	0.67%	-0.67%	-6.56%	Full Size Model, 49M Cells, 22.5 hrs, 3 body layers
		open	40%	Single Belt	0.00%	-7.19%	4.45%	Scale Model, 46M Cells, 23 hrs, 3 body layers
		closed	40%	Single Belt	1.74%	-2.48%	4.61%	Scale Model, 54.5M Cells, 25.8 hrs, 3 body layers
5	Sedan 2	open	100%	5 Belt	0.00%	-1.87%	-0.37%	45M Cells, 3 Body layers, 22 hrs, Open, Full Size
		blanked	100%	Fixed	1.57%	-26.38%	9.84%	
6	Sedan 3	closed	100%	Fixed	2.35%			42 M Cells, 3 Body Layers, 46 hrs, Full Size
		open	40%	Single Belt	0.32%	-2.27%	-2.27%	Perforated Plate Porous Zones, Simulated ARC tunnel Belt System
		closed	40%	Single Belt	2.03%	-1.35%	2.03%	Perforated Plate Porous Zones, Simulated ARC tunnel Belt System
7	Estate 1	open	40%	Single Belt	-0.32%	11.04%	26.62%	Scale Model In tunnel, 91.5 M Cells, 49.74 hrs, 3 body/belt layers.
8	Estate 2	open	100%	5 Belt	-0.95%	-3.81%	-3.17%	
9	Hatchback 1	open	40%	Single Belt	3.09%	7.21%	19.75%	Scale Model in Tunnel, 101.4M Cells, 25.65 hrs, 3 body/belt layers.
10	Hatchback 2	open	100%	5 Belt	2.18%	-22.55%	12.00%	
11	SUV 1	open	40%	Single Belt	0.81%	6.59%	-16.76%	Scale Model, 47.7M Cells, 13.4 hrs, 3 body layers
12	NASCAR 1		40%	Single Belt	2.22%			Scale Model, 53.4M Cells, 35.6 hrs, 3 body layers
13	NASCAR 2	open	40%	Single Belt	-1.25%	-32.67%	-10.47%	Scale Model
14	Semi-Truck 1	open	12.5%	Single Belt	0.19%			Tractor CD is 0.326/(0.320); trailer CD is 0.207/(0.214) Tunnel/(CFD)
15	Light Truck 1	open	20%%	Single Belt	-0.38%	-5.09%	-10.38%	63M Cells, ARC Moving Belt on Floor

Average Error Magnitude 1.2% 9.37% 9.24%

Validation Process | Results Trucks

Vehicle No.	Vehicle Model	Grille (open, closed, blanked)	Yaw	Wind Tunnel Data		Elements			Notes (no. of cells, prism layers, run length, etc.)
				Scale	Ground Simulation	Coefficients			
						CD	CLF	CLR	
21	Semi-Truck 2	open	0.000	0.125	Single Belt	0.19%			Originial tractor trailer mesh settings
		open	0.000	0.125	Single Belt	-1.13%			updated mesh settings
		open	6.000	0.125	Single Belt	2.63%			updated mesh settings
		open	6.000	0.125	Single Belt	3.45%			
		open	6.000	0.125	Single Belt	3.45%			updated mesh settings, wider wakeboxes
		open	0.000	0.125	Single Belt	2.25%			updated mesh settings, wider wakeboxes
		open	-6.000	0.125	Single Belt	4.85%			updated mesh settings, wider wakeboxes
		open	9.000	0.125	Single Belt	2.50%			updated mesh settings, wider wakeboxes
		open	-9.000	0.125	Single Belt	3.13%			updated mesh settings, wider wakeboxes
		open	9.000	0.125	Single Belt	0.00%			updated mesh settings, wider wakeboxes
		open	9.000	0.125	Single Belt	2.06%			
22	Light Truck 2	open	0.000	0.125	Single Belt	-0.38%	-5.09%	-10.38%	Originial tractor trailer mesh settings
		open	6.000	0.125	Single Belt	2.69%	-0.36%	-17.95%	Originial tractor trailer mesh settings
		open	3.000	0.125	Single Belt	1.47%	-3.87%	14.73%	Originial tractor trailer mesh settings
23	Semi-Truck 3	open	0.000	0.125	Single Belt	0.57%			
		open	3.000	0.125	Single Belt	4.13%			
		open	6.000	0.125	Single Belt	4.98%			
		open	9.000	0.125	Single Belt	1.11%			
24	Semi-Truck 4	open	0.000	0.125	Single Belt	5.86%			

Average Error Magnitude	2.5%	3.1%	14.4%

Example | CRF / FIAT

FIAT Bravo: CFD Aero Analysis

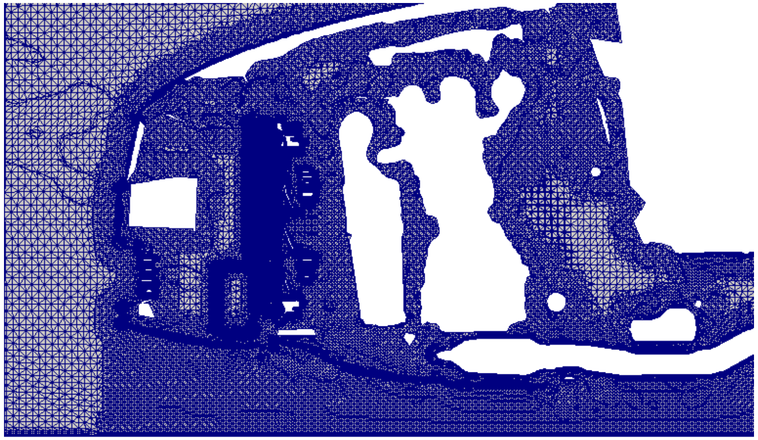
- Open cooling geometry
- Open road conditions
- Verify best CFD simulation practices for accurate prediction of drag/lift
- Comparison to wind tunnel and commercial CFD code (not shown here)

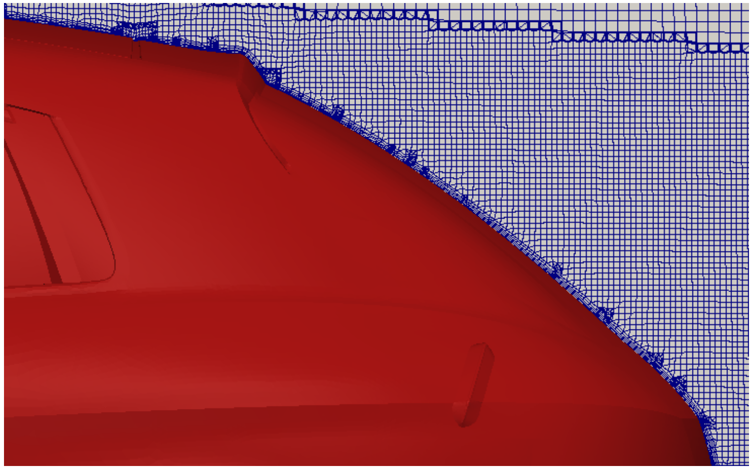
With kind permission of CRF

Example | CRF / FIAT

FIAT Bravo: CFD Mesh

- Automatic hex-based dominant mesh created in ELEMENTS → 43.5M cells
- Localised volume refinements to resolve wake and other important areas
- Near wall layers to model boundary layer flow
- Zonal meshing to capture porous regions





Example | CRF / FIAT

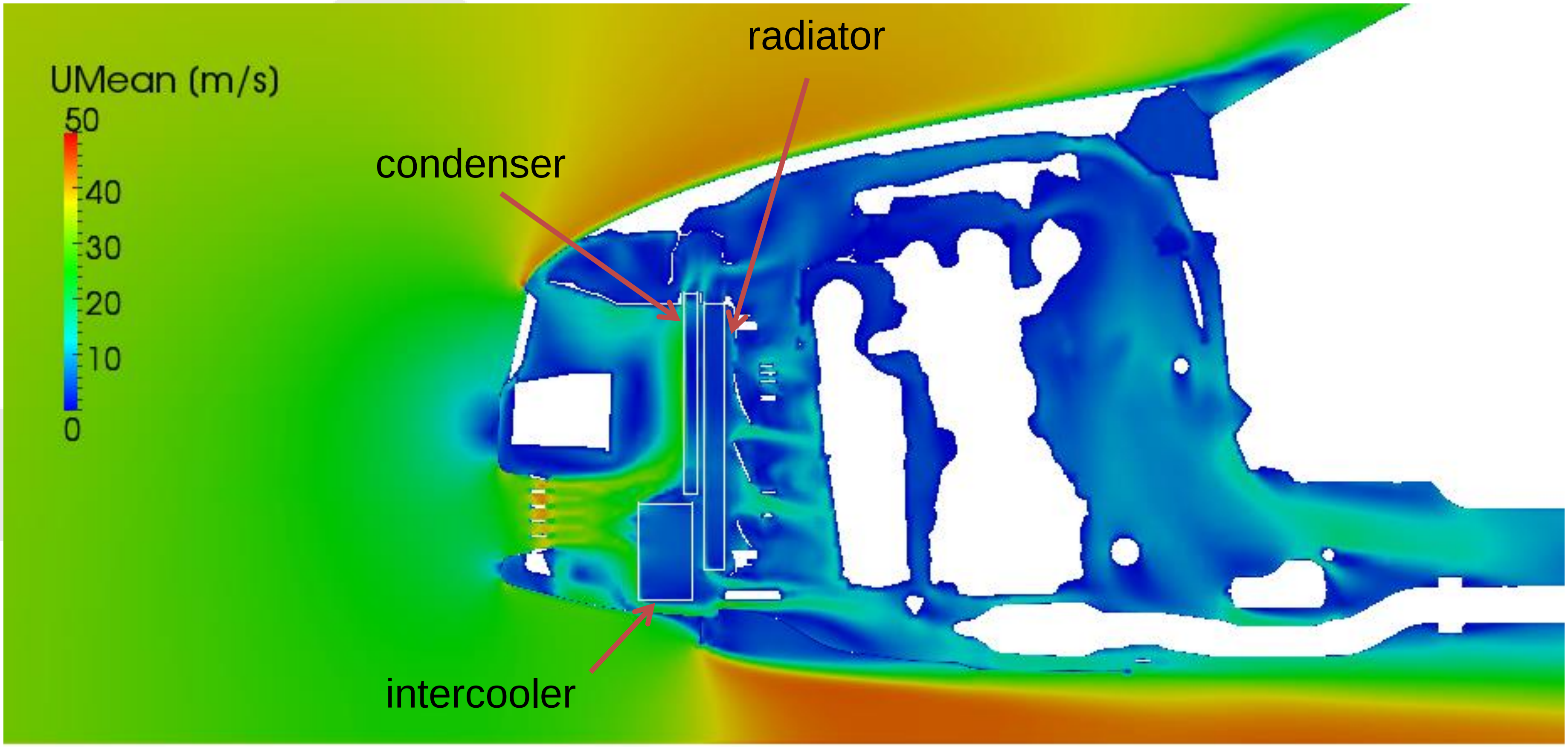
FIAT Bravo: CFD Setup

- Transient incompressible flow analysis
- DDES – Spalart Allmaras
- Velocity → 140 km/h (38.89 m/s)
- Porous modelling: condenser, radiator & intercooler
  - Darcy-Forchheimer
- Moving ground with rotating wheels (moving walls)
- Ignore thermal effects (cold flow)

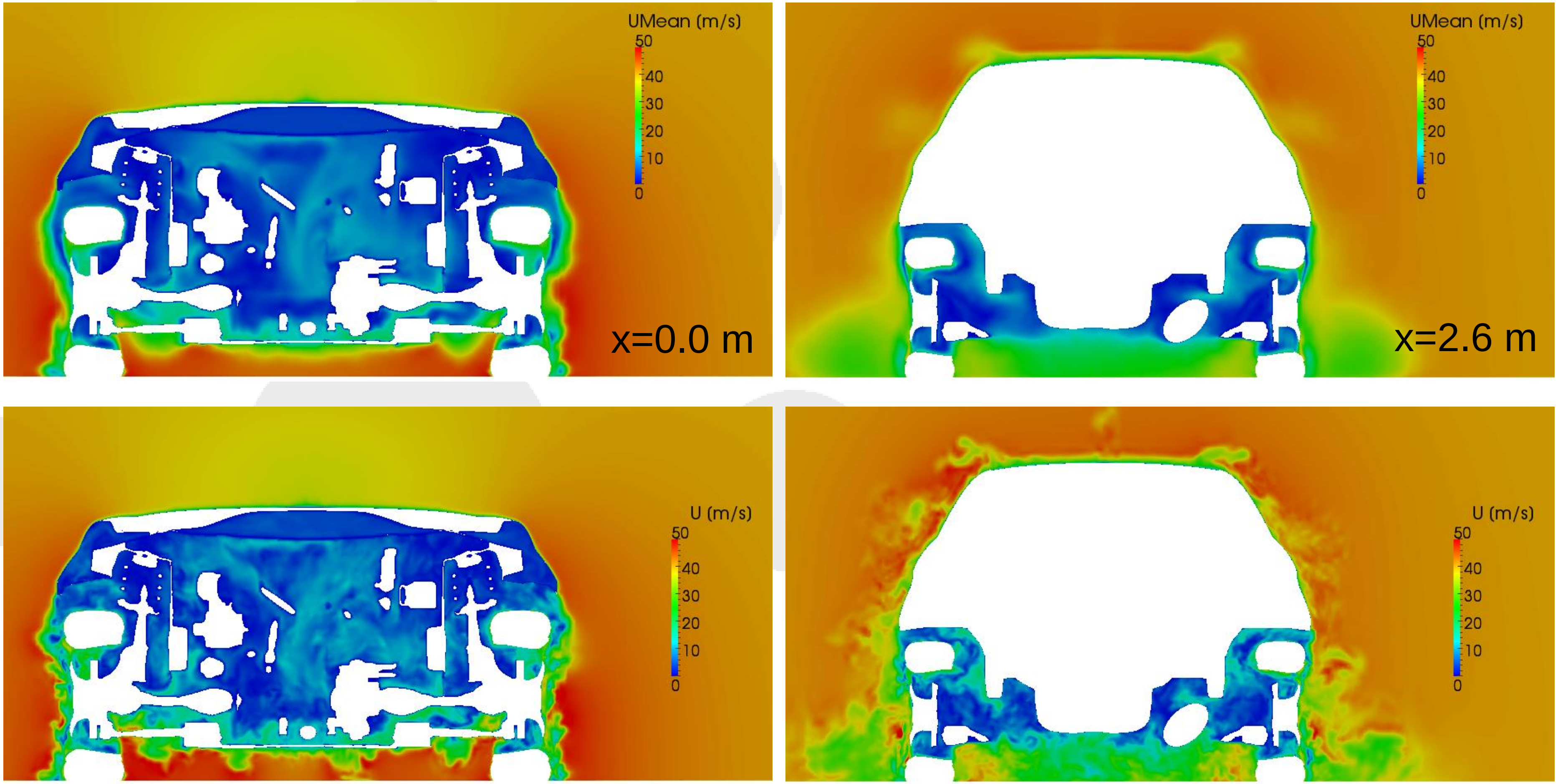
Example | CRF / FIAT

FIAT Bravo: Mean Pressure

FIAT Bravo: Underhood Velocities

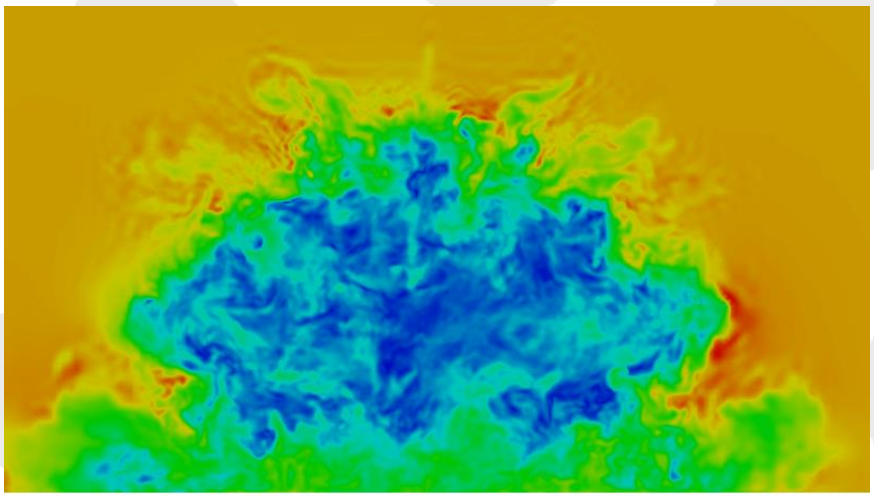


FIAT Bravo: Wheel Plane Velocities

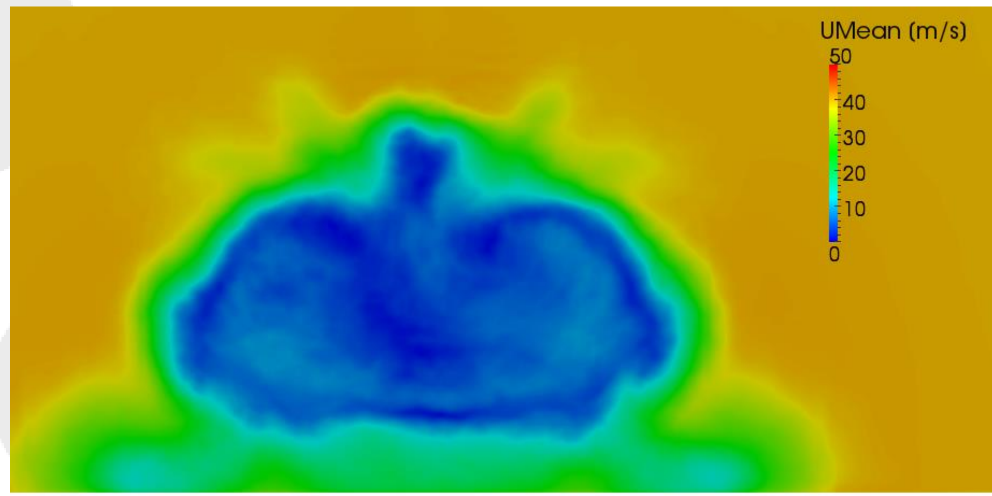


FIAT Bravo: Wake Velocities x=3.4m

Instantaneous

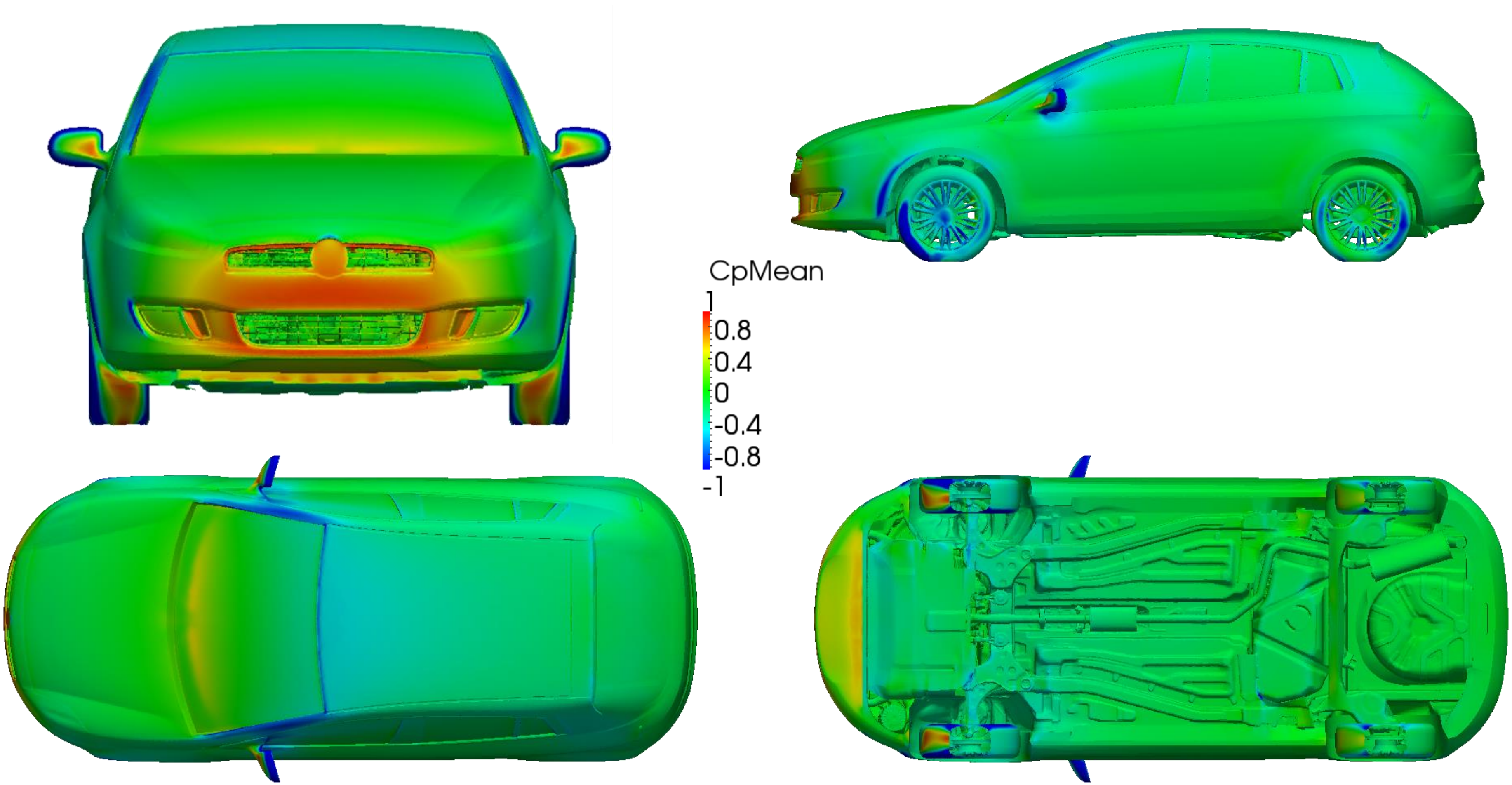


Mean



Example | CRF / FIAT

FIAT Bravo: Mean Pressure Coefficient



FIAT Bravo: Wall Shear Stress

Instantaneous

Mean

Low
High

Example | CRF / FIAT

Main Benefits for Customer

- Considerable cost savings in software licenses without compromising quality of results or ease of use (in comparison to existing CFD provider)
- Increased scalability without license constraints
- Maximum utilisation of in-house HPC resources
- Reduced CAD preparation and turn-around times
- Universal open-source CFD data platform for exchanging and visualising results
- Consistent use of best simulation practices across all departments and brands

Concluding Remarks

- Developed method proven to be fast, stable, reproducible, and efficient
- Meshing approach guarantees high quality meshes
- Flow modelling methods leverages DDES turbulence
- Blended advection schemes captures relevant flow phenomena
- Time-step adjustment schemes decrease overall computational time
- Over 100 vehicle setups validated to develop “best practices”
- Drag prediction within 1.2% of experimental results
  for passenger vehicles and 2.5% for trucks

Concluding Remarks

Challenges

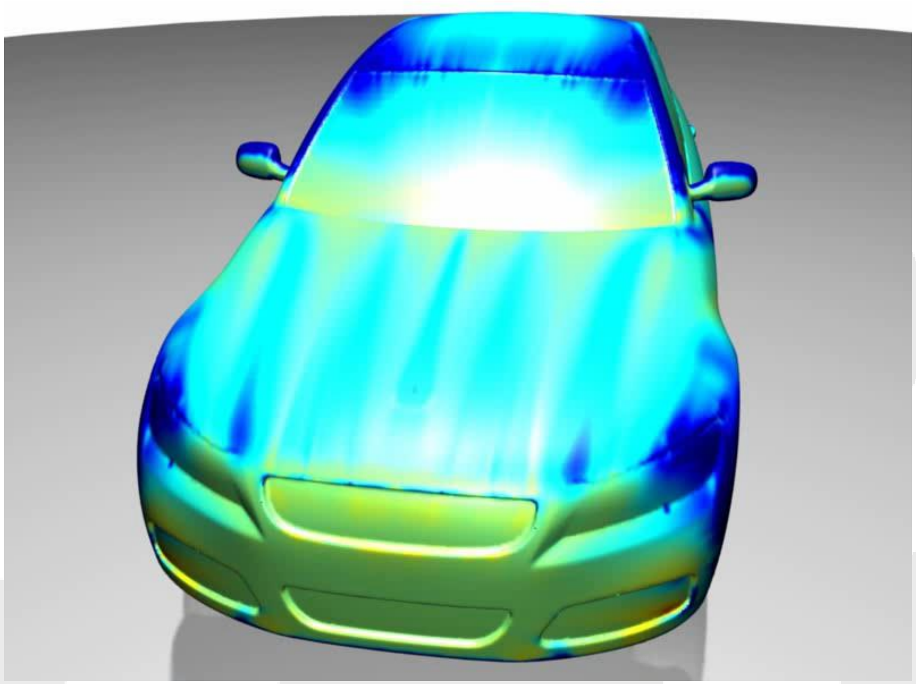
- Wheel modelling
  - Account for induced drag (pump effect)
  - Grooves and tyre deformation
  - Large impact on cooling drag and lift prediction
  - Current methods (MRF, surface velocity) have shortcomings
  - Sliding mesh currently too computationally expensive
- Significant variance in results based on initial conditions
  - Decomposition, initial conditions, solution profile, small geometric changes
  - Very long time averages requires to eliminate stochastic element
  - Poses problems for design applications

engys

Future Developments

- Continuous modelling improvement
  - DDES → IDDES, porosity, solution algorithm
- Automated adaptive meshing
  - Gradient analysis (p & U)
  - Surface resolution adequacy (y+, P+)
  - LES SGS energy ratio
- Improved accuracy for larger time-step sizes
  - Time discretisation (stable 2.5th order possible)
  - Inclusion of dt*velocity scale in LES length scale
- Scalable AMI
  - Rotating wheels

Further Developments



- Shape and topology optimisation
- Unique continuous adjoint formulation
- Fully 2nd order accurate
- Surface morphing driven by user-specified objectives
- Steady-state using “frozen” turbulence method
- DDES time-averaged primal solution

Questions?

THANK YOU!