Complex Flow Problems

An Application of Empirical Design, CFD, Test and Optimisation Methods

Clive Hudson

Eclipse Sports Cars

Agenda

- Background of the Project & Empirical Flow Design
- A CFD Steady State Assessment
- Diffuser Development through Design of Experiments
- CFD / FEA Coupling – Researching what's really happening
  - The method
  - Current application to the Eclipse SM1
  - Possible applications
- Summary and Conclusions

Background of the Project & Empirical Flow Design



Background

- *Why build a new car?*
- Available cars all seen as compromised in their design
- To see if we're right
- For the learning experience
- Because we can…

The Project

- Racing allows a very focussed project
- Timescale – 9 months to Autosport show
- Design brief
  - Build at least 5 cars
  - Driver friendly
  - Low aero and low power
  - Safe and repairable
  - Cost

9 Intensive Months

- Many decisions to be made quickly at the start, and many more along the way
- Key features:
  - Transverse engine and gearbox behind the cockpit, with a chain drive from each driveshaft to the rear wheels – for weight distribution, low inertia, suspension packaging, diffuser
  - Laser-cut mig-welded steel tube chassis, for which we'll have to make the jigs
  - 100mm of suspension travel – long double wishbones at the front with pushrods, and a super stiff but light de Dion 'trellis' at the rear



Mostly a very satisfying project

- Design and build of first car overlap as far as possible to meet timescale
    - Labour resource very limited
    - More bespoke machined components than originally planned
    - Body design in Rhino3d
    - Everything else and FEA in Creo
- Fibreglass body buck and moulds manufactured by traditional methods as the only route to achieve balance of cost and speed

Empirical aero design

- Philosophy – don't mess it up!
- No tools or time to develop anything complicated
- Low drag, avoid creating lift, keep an eye on stability
- Key features:
  - Minimise frontal area, simple, and relatively narrow
  - Flat floor, leaving space for rear diffuser
  - Adequate cooling (radiator, engine bay, brakes)
- Maintain options for future developments
- Aesthetics are a significant influence – drivers, teams, and sponsors want to win, but not in an ugly car
- 'Ignoring' downforce allows complete focus on mechanical grip and driveability

Very rewarding to drive

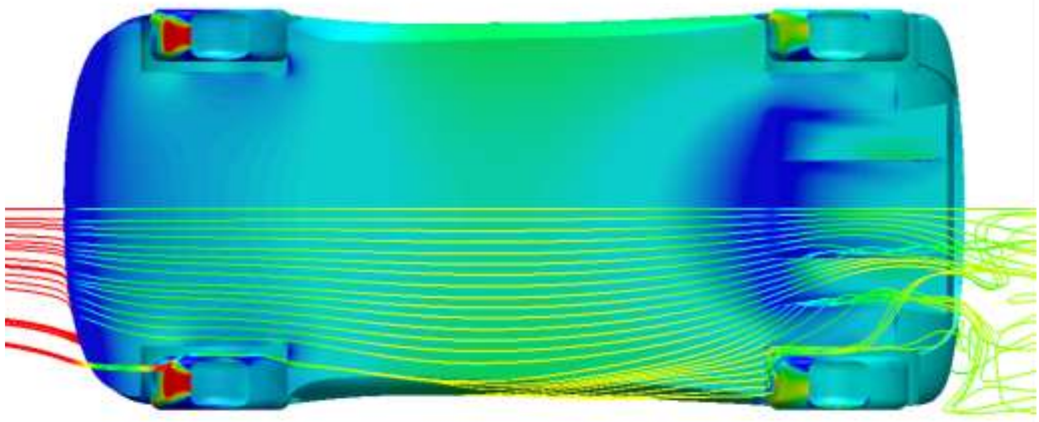
12 lap records

Won every race in 2017

Aero development

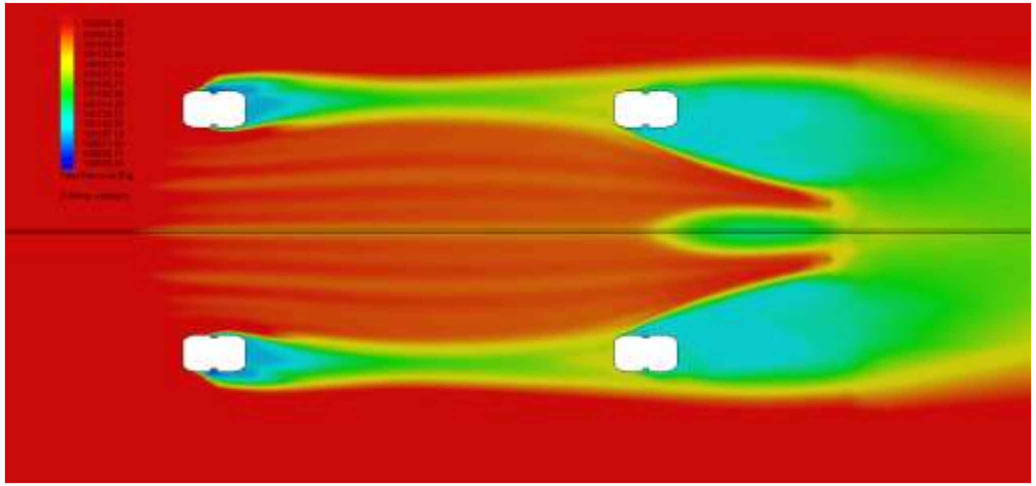
- Great success so far
- *What can we do with aero development*
- Rear diffuser is obvious first target – downforce at the rear would help, and may achieve a reduction in drag
- Empirical diffusers tried but without obvious benefit
- Time to turn to CFD – but you can't do useful CFD in your spare time…

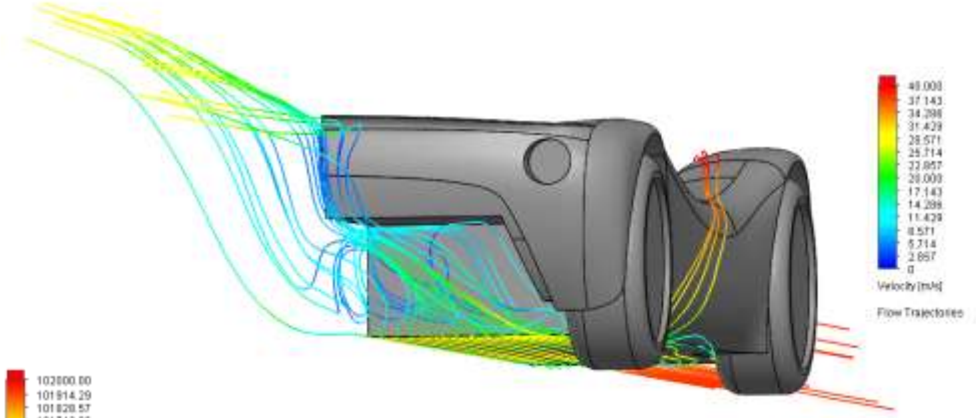
Steady State CFD Assessment and Diffuser Development

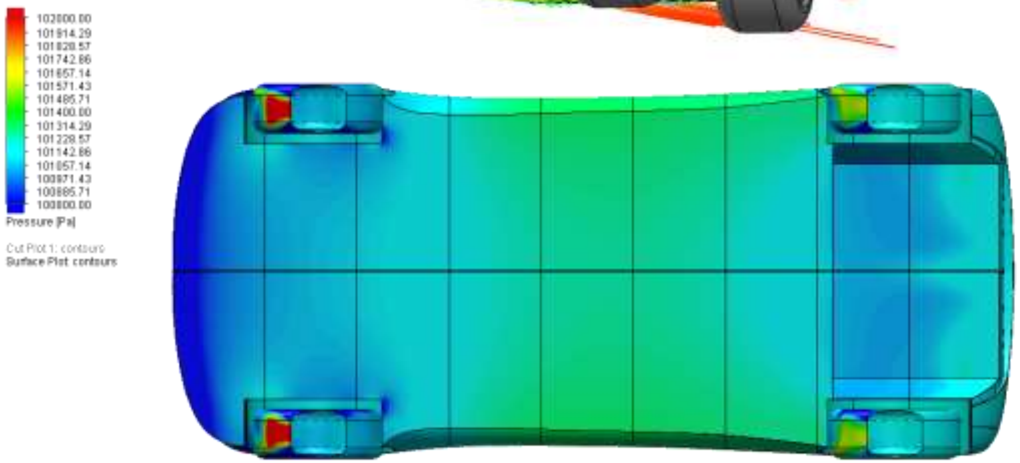


Baseline Results (Open Rear End)

Vehicle Velocity = 80mph







(Above): Plotting Total Pressure 40mm above the ground plane, shows the circulation about the car's rear. The same effect can be seen in the car's streamlines. (Above Right)

Areas of low pressure indicate where downforce is generated on the surface plot right. Currently most is created by the car's existing splitter.

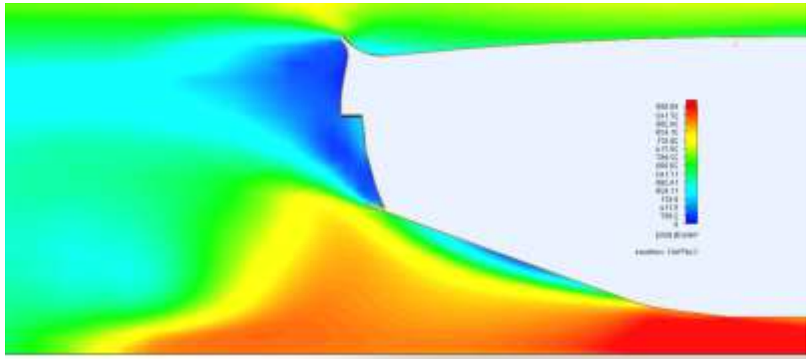
Car Drag	400 N
Car Downforce	300 N

Existing Diffuser Studies

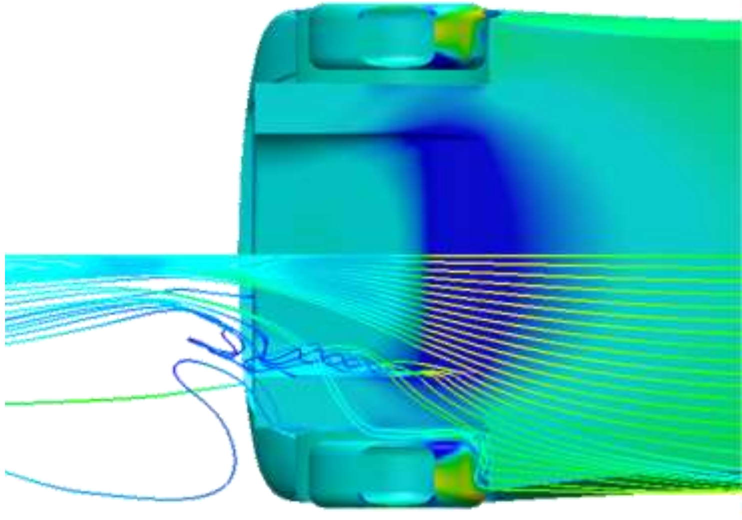
- Pre-CFD, two diffusers were developed
  - one conservative
  - one aggressive

- Designs assessed using SOLIDWORKS Flow Simulation CFD solver

- How did they perform on the car?



Flow Velocity



Surface Pressure &
Flow trajectory

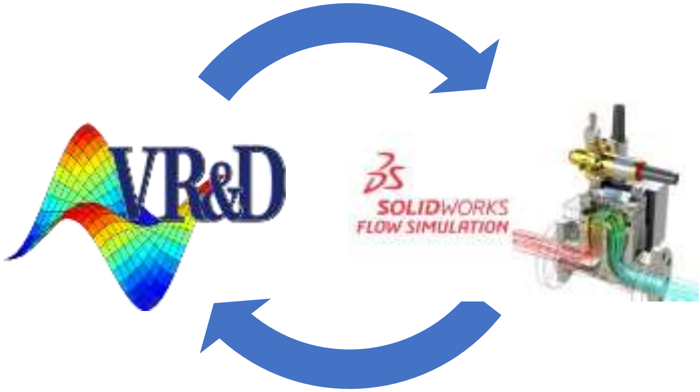
	Drag	Downforce
Base Car	400N	300N
7˚ Diffuser	380N	318N
20˚ Diffuser	390N	440N

VisualDOC Parametrised Optimisation

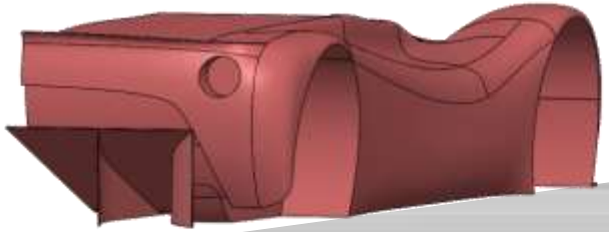
With the idea to create an optimised design, a parametrised diffuser was created and paired to a DoE study built using Vanderplaats VisualDOC.

The design was built with 4 varying parameters:

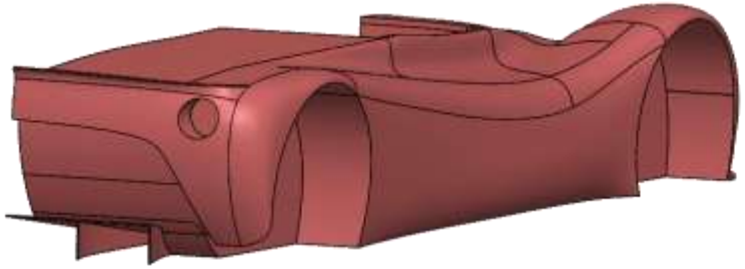
- Exit Width
- Diffuser Length
- Diffuser Angle
- Rear Ride Height (Rake)



Maximum and Minimum dimensions bound



A total of 25 different designs were generated for simulation



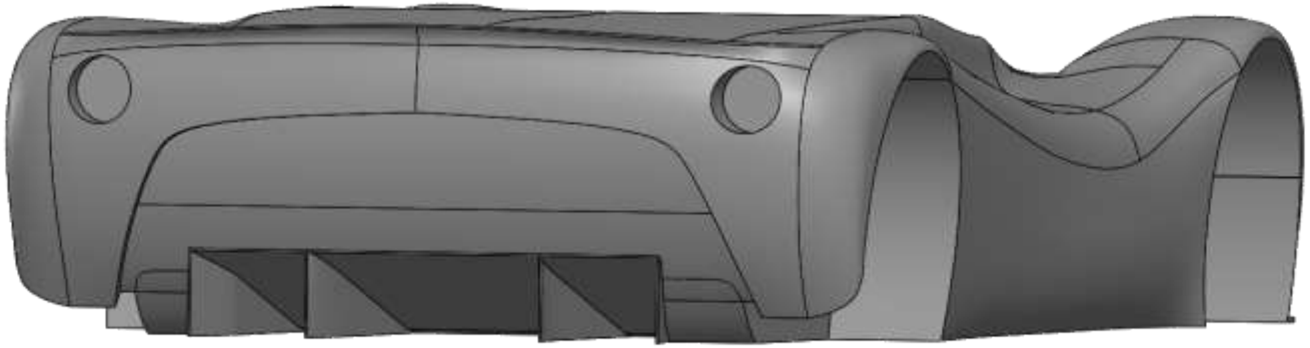
Diffuser geometry generated using VisualDOC imported into CAD as a design table

VisualDOC Optimised Design

We are also able optimise within VisualDOC to generate optimum geometry dimensions to create the diffuser that theoretically will create the most downforce.

Diffuser Angle	12.3^{o}
Diffuser Length	710 mm
Diffuser Width	432 mm
Rear Ride Height	79.4 mm
Car Predicted Drag	**400 N**
Car Predicted Downforce	**540 N**

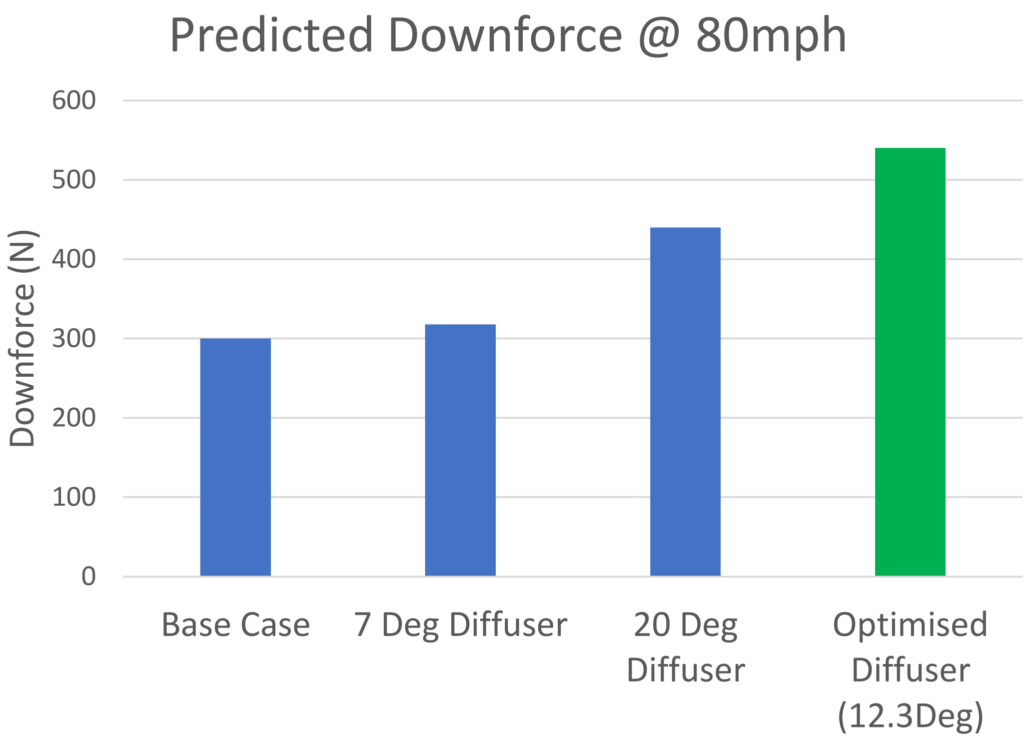
Optimised Diffuser Design



The optimised design features the smallest length possible with a small amount of rake and a medium angle of 12.3 degrees.

Summary of Diffuser Studies

- Empirical not bad at all
- Combining CFD with DoE/Optimisation has yielded a predicted further improvement
- Is it real?... how does driving the car influence things?



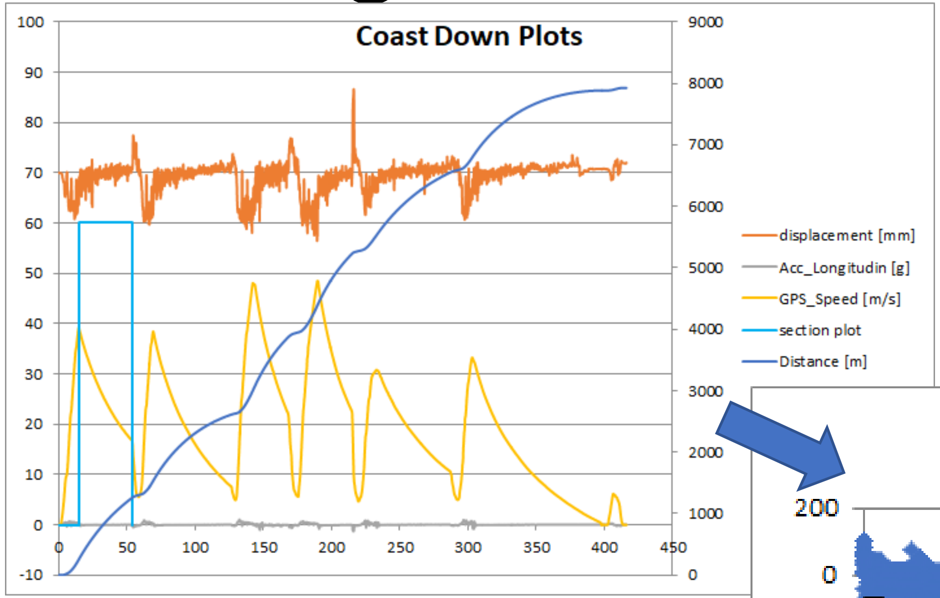
Validation

Steady-State & Coastdown Testing

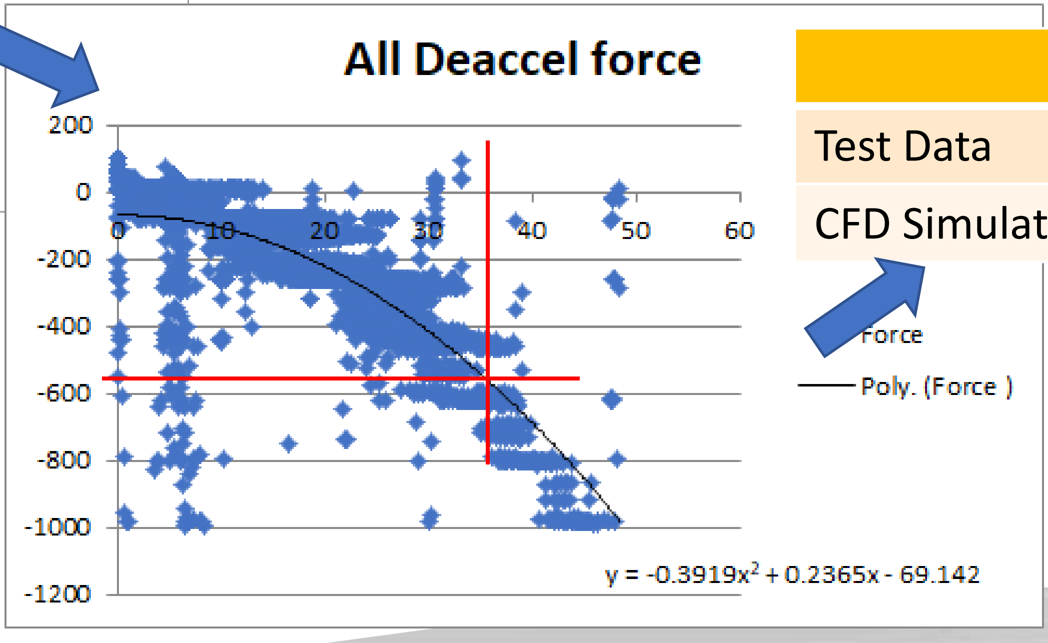
Bruntingthorpe’s ‘Motorway’ section was used to undertake aerodynamic testing of car with manufactured diffuser

Pressure testing instrumentation developed and assembled by GRM

Drag Calculations & Comparison



- Reasonable correlation
- Difference attributed to
  - No internal flow (radiator cooling)
  - No driver
  - No roll-bar

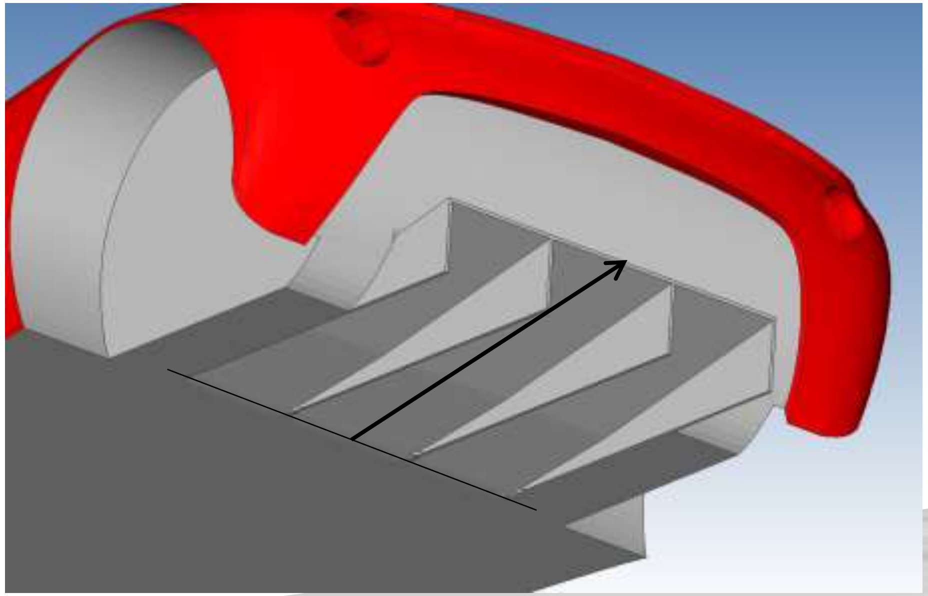


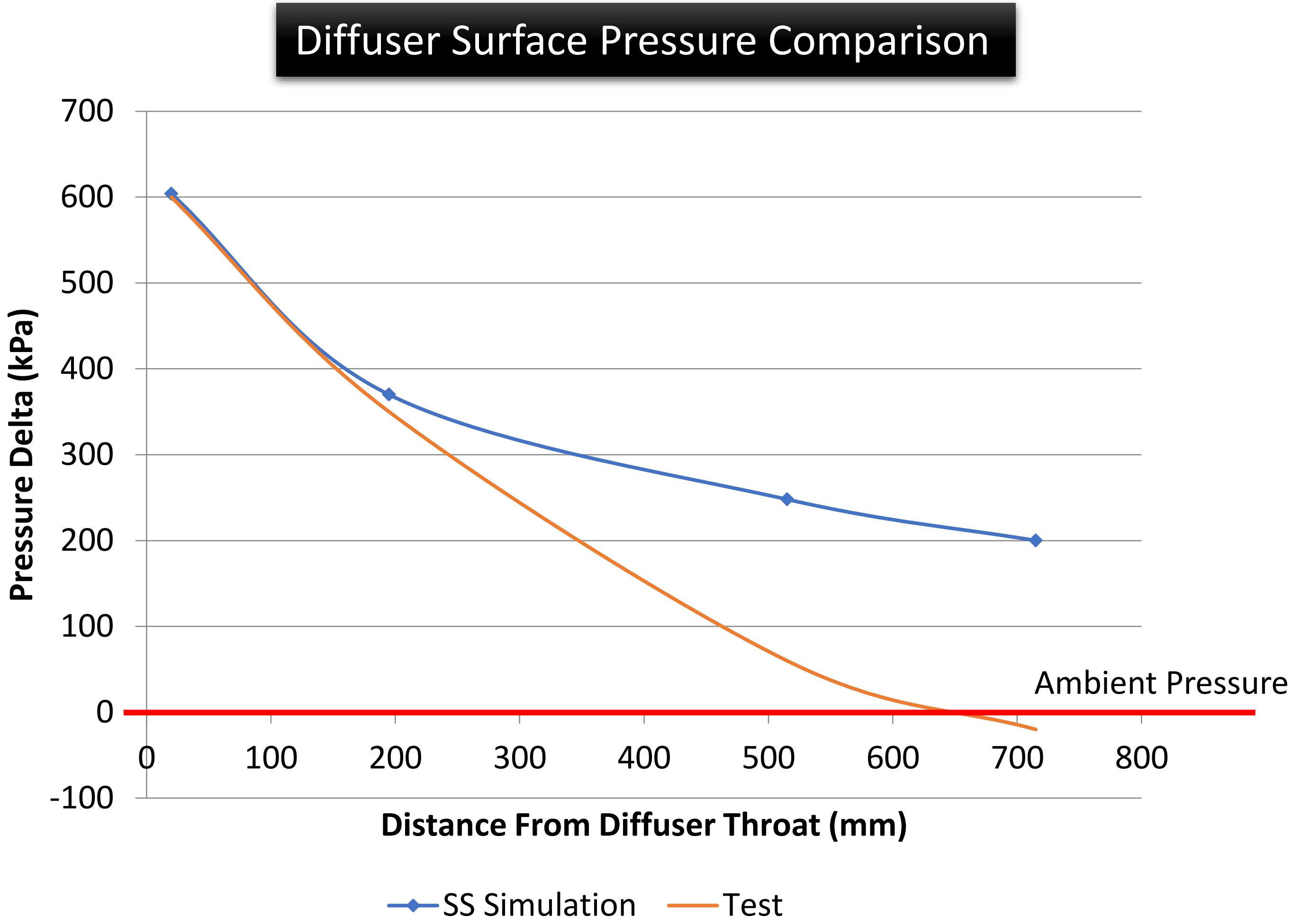
	Drag
Test Data	550N
CFD Simulation	400N

Diffuser Pressure Comparisons

Throat and first stage of diffuser shows extremely good correlation between simulation predictions and test

Test data shows significant reduction in surface pressure in second stage of diffuser, indicating flow separation





Transient Flow / Structure Coupling

Objective

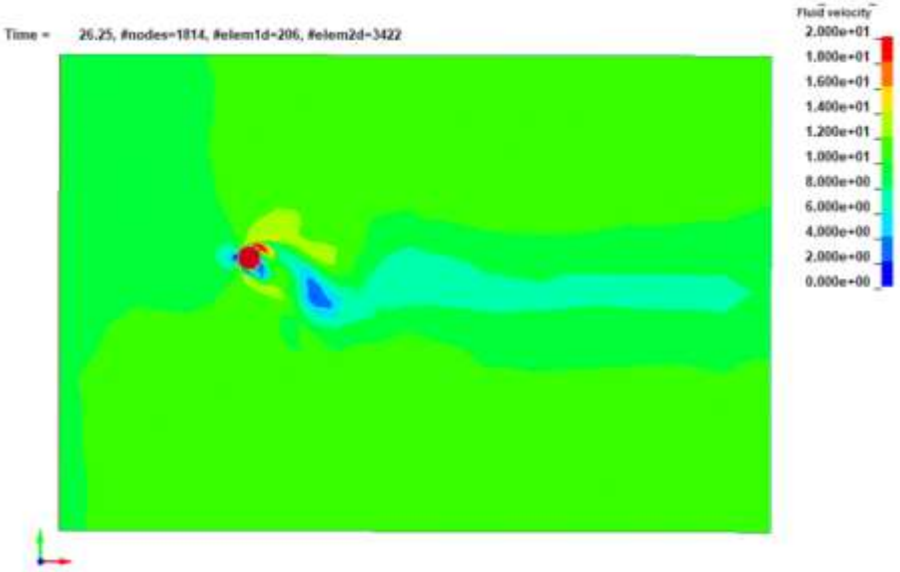
- Working with GRM, an ambition for R&D activities is to develop a method of studying the influence of vehicle motion on flow behaviour

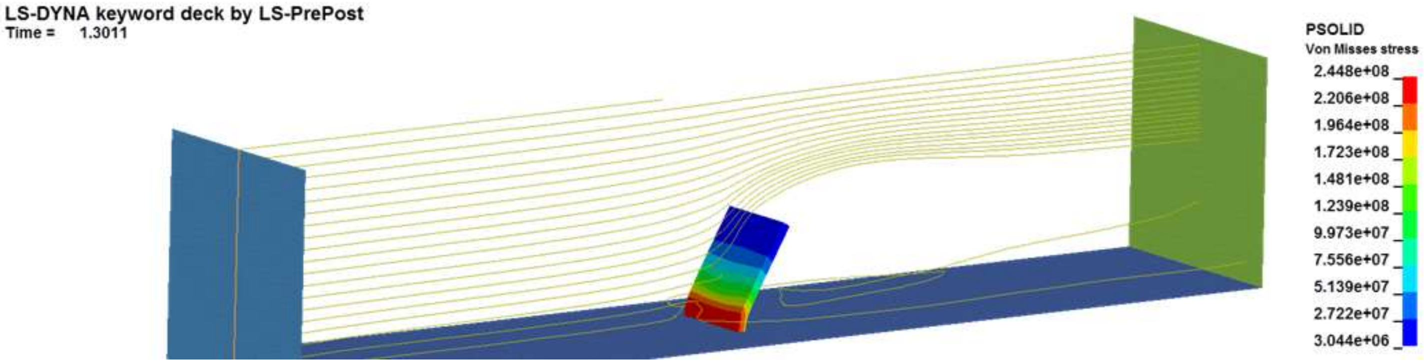
The Approach

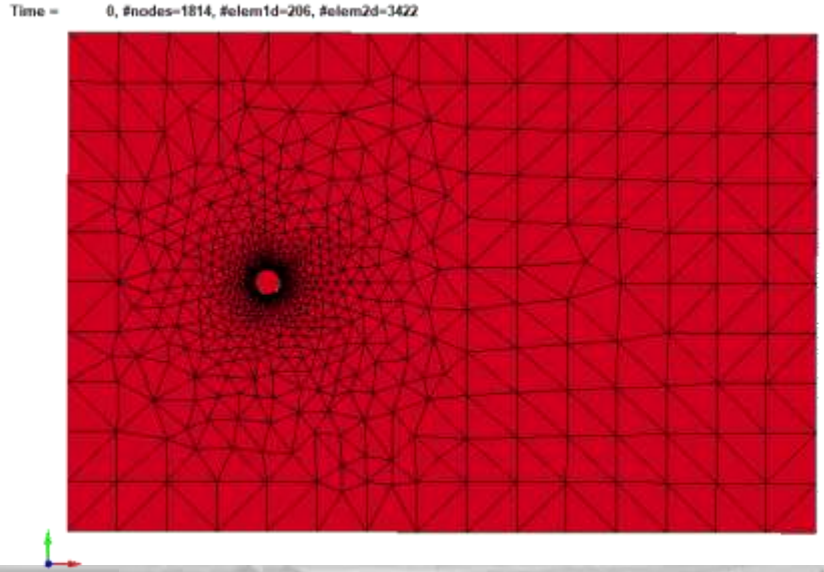
LS-DYNA contains a transient ICFD solver

Solution fully couples to FEA solver

Objective to develop an understanding of these capabilities





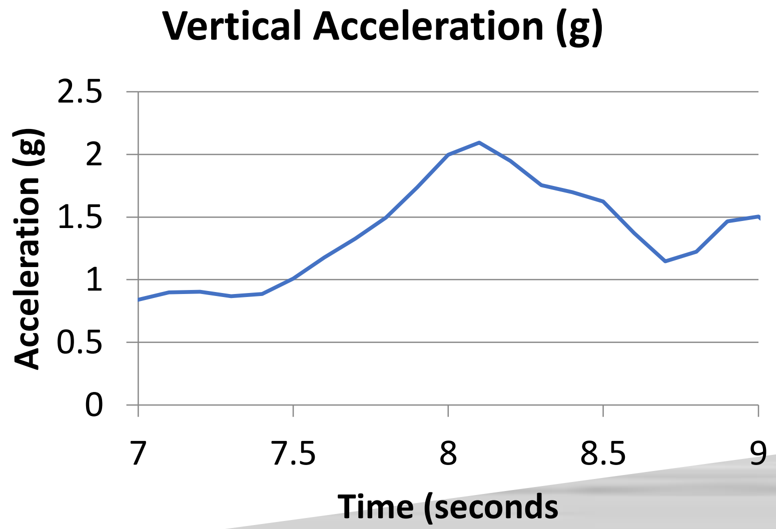


An Example Case

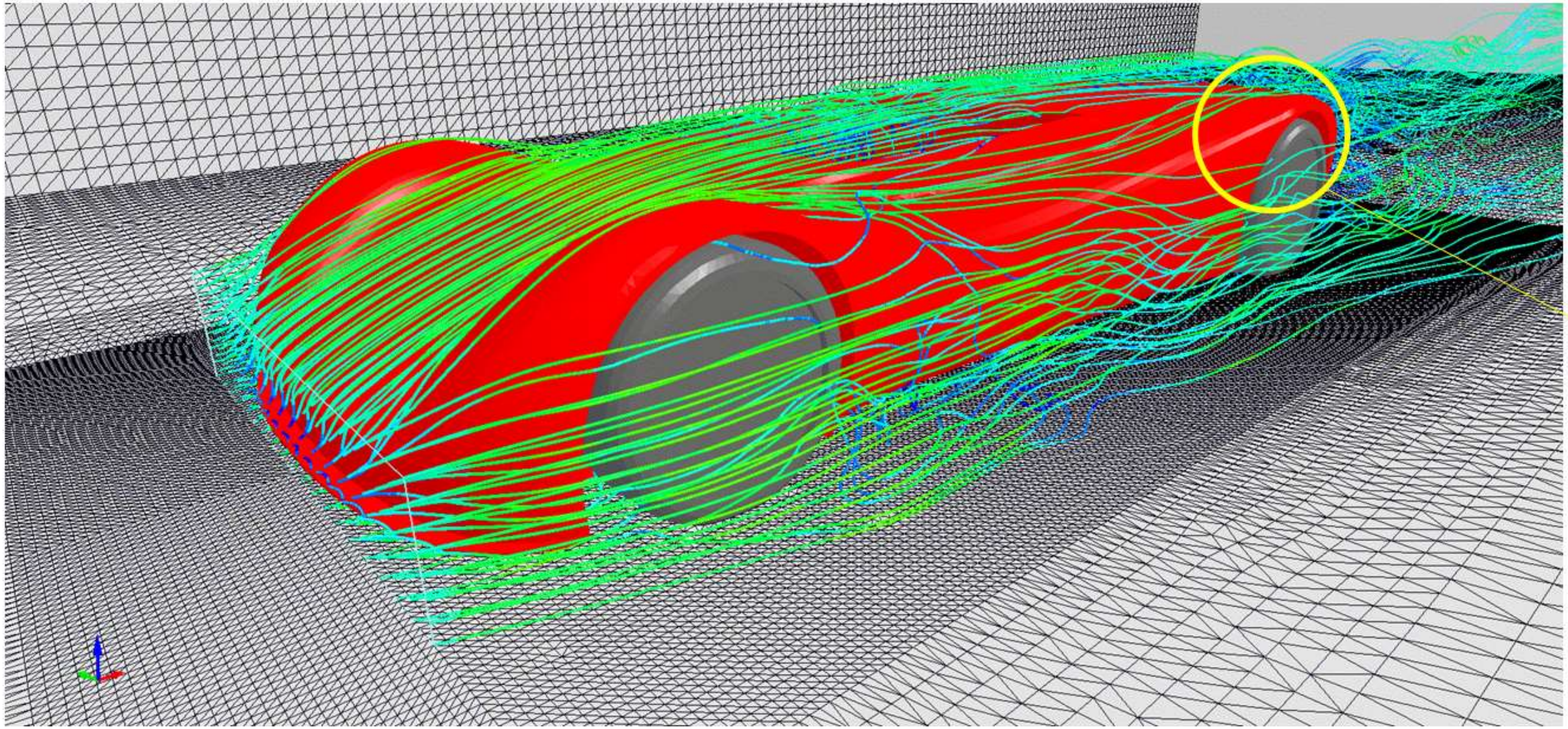
Paddock Hill bend at Brands Hatch

Elevation change causes a considerable vertical acceleration, causing car ride height to reduce significantly

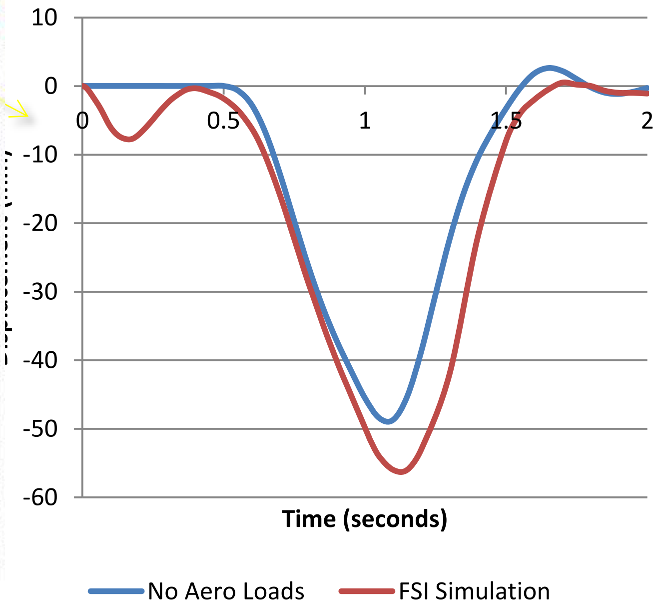
What happens to the aero flow if we apply this vertical acceleration?



Coupled CFD / FEA Simulation



- Dynamic ride height change of car increases effectiveness of diffuser, further reducing ride height during event



Summary & Conclusions

- Traditional Steady State method proven to be a reliable approach
- Allows application of DoE and optimisation techniques
- Ambition for transient, Coupled flow methods:
  - For Eclipse
    - Develop more insight into real on-track flow behaviour
    - Eliminate need for multiple steady-state investigations
  - Other Applications
    - Lots…
    - Automotive applications - Bonnet flutter, wheel arch & underfloor trim
    - F1
    - America’s Cup
    - Renewable Power Generation