



## **CHAM Limited**

***Pioneering CFD Software for Education & Industry***

Bakery House, 40 High Street, Wimbledon Village, London SW19 5AU, UK  
Tel: +44 (0)20 8947 7651 Fax: +44 (0)20 8879 3497 Email: [phoenics@cham.co.uk](mailto:phoenics@cham.co.uk)  
Web: <http://www.cham.co.uk>

### **F1 VWT Formula 1 Virtual Wind Tunnel MkV September 2010**

**CFD software prepared for  
F1 in Schools Challenge**

F1 VWT is a variant of CHAM's PHOENICS CFD product. F1 VWT was created for use exclusively within schools and colleges participating in the F1 in Schools Challenge sponsored by, amongst others, BAE Systems, Denford, Jaguar.

PHOENICS is developed and owned by CHAM Ltd, Bakery House, 40 High Street, Wimbledon Village, London SW19 5AU, UK, tel: +44 (0)20 8947 7651, fax: +44 (0)20 8879 3497, email: [phoenics@cham.co.uk](mailto:phoenics@cham.co.uk), web: [www.cham.co.uk](http://www.cham.co.uk)

F1 VWT is distributed exclusively by Denford Ltd, Birds Royd, Brighouse, West Yorkshire, HD6 1NB, UK, tel: +44 (0)1484 712264, fax: +44 (0)1484 722160, email: [info@denford.co.uk](mailto:info@denford.co.uk), web: [www.denford.com](http://www.denford.com)

**Title:** F1 VWT Formula 1 Virtual Wind Tunnel  
**Document rev:** 06  
**Doc. release date:** 1<sup>st</sup> September 2010  
**Software version:** PHOENICS 2009

**Editor:** John C Ludwig  
**Published by:** CHAM  
**Other contributors:** Sebastien Combacher, Sahil Dawa & Alex Hawkins  
**Confidentiality:** N/A  
**Classification:** Unclassified

---

The copyright covers the exclusive rights to reproduction and distribution including reprints, photographic reproductions, microform or any other reproductions of similar nature, and translations. No part of this publication may be reproduced, stored in a retrieval system or transmitted in any form or by any means, electronic, electrostatic, magnetic tape, mechanical, photocopying, recording or otherwise, without permission in writing from the copyright holder.

© Copyright Concentration, Heat and Momentum Limited 2004-2010

CHAM, Bakery House, 40 High Street, Wimbledon, London SW19 5AU, UK  
Telephone: +44 (0)20 8947 7651 Fax: +44 (0)20 8879 3497  
E-mail: [phoenics@cham.co.uk](mailto:phoenics@cham.co.uk)  
Web site: <http://www.cham.co.uk>

## Preface

The aerodynamics of racing cars has always been very important. Two specific forces are of considerable concern:

- Aerodynamic drag or air resistance
- Aerodynamic lift which can reduce or enhance traction or handling of the car

F1 teams expend a lot of effort using aerodynamic prediction tools to achieve a car body that has low drag and sufficient negative lift (down force) to give good handling,. In the past, wind tunnels were the only tool used for this task, but in the last 10 years Computational Fluid Dynamics (CFD) has enabled good predictions of lift and drag at an early design phase. PHOENICS is such a CFD tool. CFD is the simulation of any process involving fluid flow using mathematical techniques embodied within a computer program. Access the Web and browse through "What is CFD" – [http://www.cham.co.uk/phoenics/d\\_polis/d\\_info/cfdintro.htm](http://www.cham.co.uk/phoenics/d_polis/d_info/cfdintro.htm) for background information.

The purpose of the F1 VWT (Virtual Wind Tunnel) subset of PHOENICS is to enable the testing of prototype vehicles before construction. This allows the designer to create and virtually test multiple shapes and styles, and achieve an optimum design before physical testing in a real wind tunnel.

The F1 VWT can be run on any 'standard' PC running Windows. However, the software can be both CPU- and RAM-intensive, so the more power there is available, the better the software will perform. Consider a 2.4GHz system, with 1Gb RAM as a suitable minimum.

The F1 VWT has been set up to accept designs created using ProDesktop, and indeed other CAD software exporting STL (3D object-based solid models) data. The default view is the unmodified balsa-wood block supplied with the F1 in Schools Kit, which may be replaced by new prototype designs imported from CAD. Designs may be for D-class or R-class bodies, updated to *F1 Bloodhound* and *Formula 1* bodies in this revision.

The F1 VWT's VR-Editor uses an interactive environment whereby users can visualize and modify their racing car designs, and provides some insight into the look and feel of the user environment utilised by design engineers in industry. The F1 VWT interface has been cut-down, offering reduced options for pre-and post-processing, though the standard PHOENICS "Main Menu" is underlying. This should not be accessed as part of the F1 in Schools Challenge and only attempted by those having access to PHOENICS-2010 User Documentation.

See: [http://www.cham.co.uk/phoenics/d\\_polis/d\\_docs/introdoc/introdoc.htm](http://www.cham.co.uk/phoenics/d_polis/d_docs/introdoc/introdoc.htm)

For reasons of simplification and computational efficiency, the default settings in the F1 VWT's VR-Editor include an inlet, an outlet, a pre-defined air-flow rate, plus pre-set structured computational grid (mesh) settings. The wheels, though spoked, are transparently 'blocked' for the purposes of the default analysis. In order to emulate the conditions anticipated within the F1 in Schools physical wind tunnel, the wheels are not rotating and the CO<sub>2</sub> cylinder produces no thrust.

Having set up and run cases, the users can visualize the results within the F1 VWT's VR-Viewer. This will include vectors, contours, iso-surfaces, and streamlines for both velocity and pressure. The forces on the vehicle body will be calculated to produce drag/lift information.

Three settings of computational meshing are offered - coarse, medium and fine - the finer the mesh the more accurate the results, but the longer the computer's run time. Please experiment with these settings whilst setting up the Tutorial and Demonstration cases in order to establish practical solutions within the time available to you. For example, "Fine" mesh runs can be left 'overnight' and the results viewed in the morning. Don't forget, if necessary, the 'solver' can be interrupted at any time and the results viewed, even before a converged solution is achieved.

Whilst the F1 VWT software has "unsupported" status, questions relating to its use can be emailed to: [f1\\_vwt\\_help@cham.co.uk](mailto:f1_vwt_help@cham.co.uk) where you will be referred to your nearest support centre.

Reference documentation for the CFD enthusiast:

TR001: PHOENICS Overview  
TR006: What's New in PHOENICS-2010  
TR110: Installation of PHOENICS-2010  
TR324: Starting with PHOENICS-VR  
TR326: PHOENICS-VR Reference Guide

All of these can be accessed from <http://www.cham.co.uk/ChmSupport/tr000.php>.

***Our thanks are given to Mr Fred Stillwell, East Cobb Middle School, Marietta, Georgia, USA for his enthusiasm, advice and provision of updated Formula 1 example cases. Thanks too, to Mr Sebastien Combacher, graduate of Warwick University for his assistance in updating this document for the Mk V revision, together with students from King's College School, Wimbledon, Mr Sahil Dawa and Mr Alex Hawkins.***

## **F1 IN SCHOOLS**

The F1 in schools challenge is a competition, open to all UK-based secondary schools and colleges, in which students design and manufacture CO<sub>2</sub>-powered model racing cars. Teams of students design 3D CAD models and manufacture the CO<sub>2</sub>-powered racing cars using CNC machines. Students produce their own F1 Car and battle it out on the racetrack with other students to see how well their car compares.

For more information on the F1 in schools challenge look on the website <http://www.f1inschools.com>

There are three types of racing car; the Bloodhound (previously known as the R-type), the Formula 1 (previously known as the D-type) and the GT. The main differences between them are the dimensions. The website also gives you more information about each car type and their associated design rules.

**CONTENTS LIST**

**F1 VWT OVERVIEW ..... 1**

**Pre-processor - The VR (Virtual Reality) Editor ..... 1**

    1.1 Selecting the body class .....1

    1.2. The Menu Bar.....2

    1.3 The VR Editor Handset .....3

    1.4 Domain Attributes Menu .....6

    1.5. The Probe .....8

    1.6. The Original Shape .....9

    1.7. Car Body Orientation and Size .....9

    1.8 Getting help ..... 11

**2. The Solver – Earth .....12**

    2.1 Overview ..... 12

    2.2 Assessing Convergence ..... 13

**3. The Post Processor - VR Viewer .....15**

    3.1. The Post Processor – VR Viewer Handset..... 16

**VIRTUAL WIND TUNNEL – TUTORIAL – BLOODHOUND CLASS .....17**

**1. Generation of Stereo-Lithography (STL) files. ....17**

**2. Starting with VWT .....18**

**3. Importing the geometry .....19**

**4. Setting up the VWT solver (Earth) .....20**

**5. Running the VWT's solver (Earth).....21**

**6. Viewing the VWT Results .....22**

    6.1 Velocity vectors.....23

    6.2 Hiding Objects .....25

    6.3 Transparency.....27

    6.4 Velocity Contours.....29

    6.5 Pressure Contours.....30

    6.6 Surface Contours.....32

    6.7 Streamline plots .....33

    6.8 Iso-surfaces .....35

    6.9 Macros .....36

    6.10 Slices.....37

    6.11 Minimum and Maximum Values.....38

**7. Coefficient of Drag and Aerodynamic Forces .....39**

**VIRTUAL WIND TUNNEL – TUTORIAL – FORMULA ONE CLASS .....41**

**DEMONSTRATION CASE - THE Balsa WEDGE .....45**

**1. Introduction .....45**

**2. Geometry, Setup and Running .....45**

**3. The Results.....47**


**4. Increase in aerodynamic efficiency .....51**

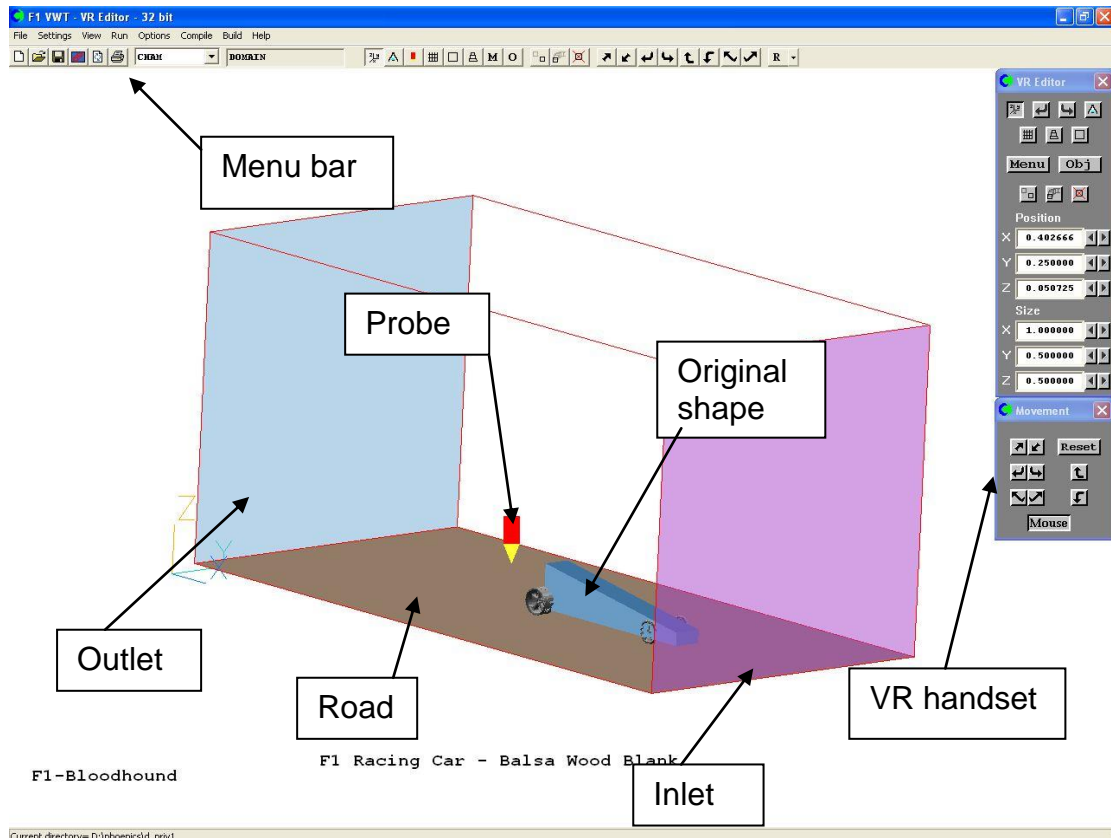
**5. The drag forces.....54**

<b>DEMONSTRATION CASE – JAGUAR F1 CAR .....</b>	<b>55</b>
<b>1. Introduction .....</b>	<b>55</b>
<b>2. The Geometry .....</b>	<b>55</b>
<b>3. The Results.....</b>	<b>56</b>
<b>4. Conclusion.....</b>	<b>61</b>

## F1 VWT Overview

### Pre-processor - The VR (Virtual Reality) Editor

The Pre-processor is the part of the CFD program that sets up the aerodynamic test on the F1 model. From the desktop, click on the **VWT icon** . Once initiated, the first screen should show a red-outlined (**Domain**) box with the default F1 balsa wood model. Everything in the **VWT** is created within this area and defines the limit of the space allocated.



#### 1.1 Selecting the body class

The body class can only be selected when starting a new case – it cannot be changed part way through a set up. To start a new case, click

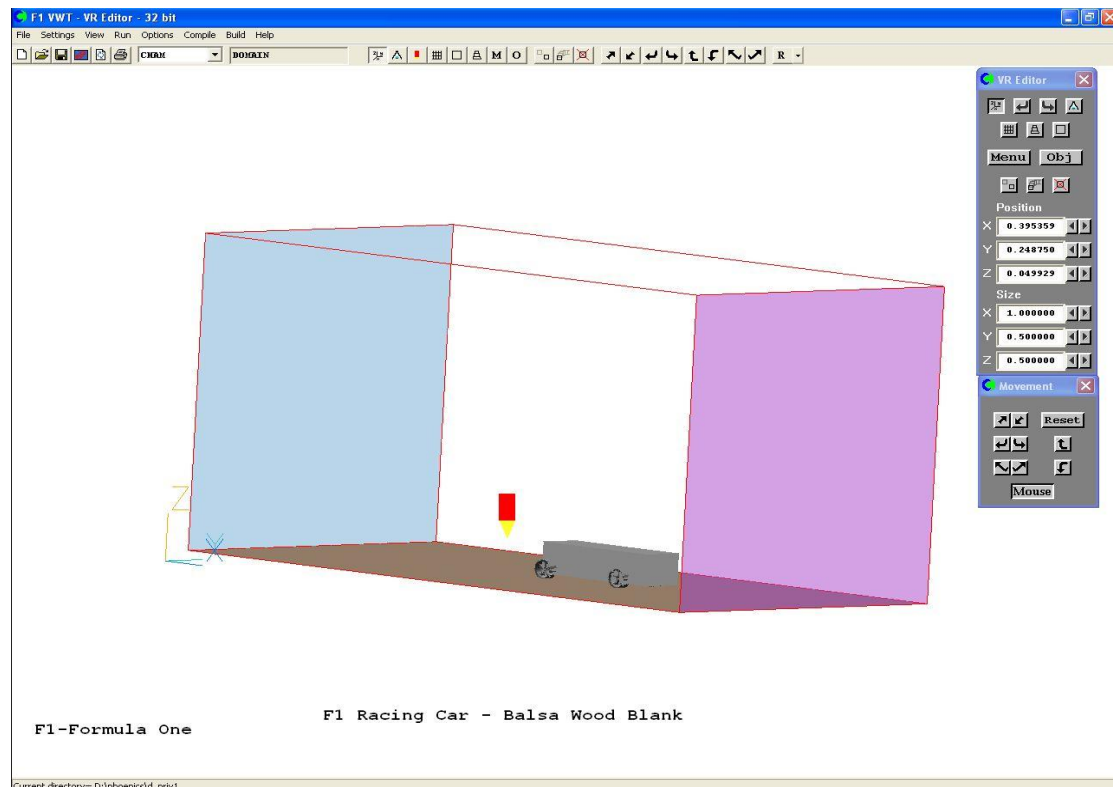
- File  
⇒ Start new case

on the menu bar. From the dialog that appears, choose F1-Bloodhound for the Bloodhound class body (formerly F1-D) or F1-FormulaOne for a Formula One class body (formerly F1-R), or F1-GT for a GT class body. The examples in this guide are for a Bloodhound class body, but the methods used are identical for all classes.

The title F1-Bloodhound in the bottom-left corner of the image indicates that this is a Bloodhound class model.



The balsa blank for the Formula One class looks like this.



Note that the title in the bottom-left corner is now F1-FormulaOne

## 1.2. The Menu Bar

This gives access to everything in the **VWT**, but there are only certain important functions needed for the F1 in Schools Challenge. Everything else can be accessed from the 'VR Handset'.

### Menu Bar

#### File...

- |                      |  |
|----------------------|--|
| ⇒ Open existing case | Allows the opening of a previous simulation                        |
| ⇒ Save as a case     | Saves the current simulation                                       |
| ⇒ Save window as     | Allows a screen shot in <b>.GIF, BMP, PCX</b> or <b>JPG</b> format |
| ⇒ Exit               | Close down <b>VWT</b>  |

#### Run...

- |                  |                         |
|------------------|-------------------------|
| ⇒ Solver         | Runs the solver         |
| ⇒ Post-processor | Runs the post-processor |
| ⇒ Pre-processor  | Runs the pre-processor  |

#### Options...

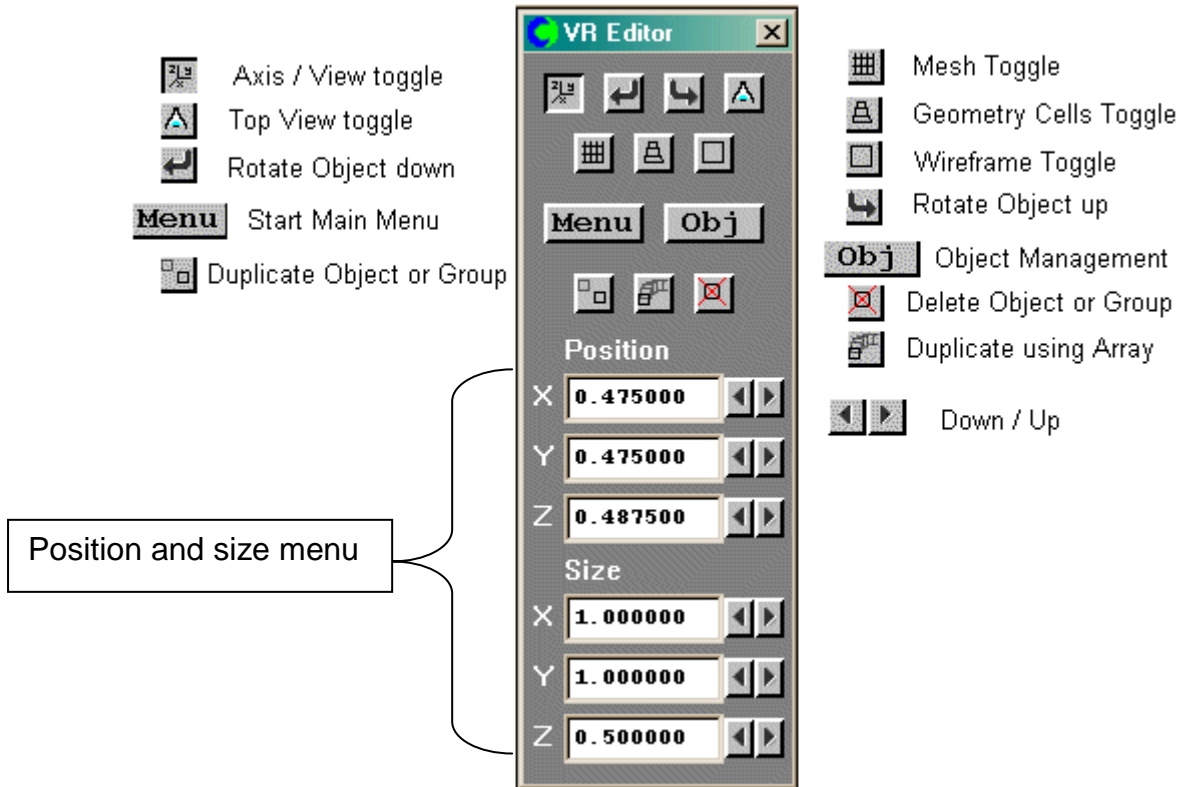
- |                            |                                 |
|----------------------------|---------------------------------|
| ⇒ Change working directory | Change to a preferred directory |
|----------------------------|---------------------------------|

Note: The other options present in the menu bar should only be investigated in the presence of a teacher.

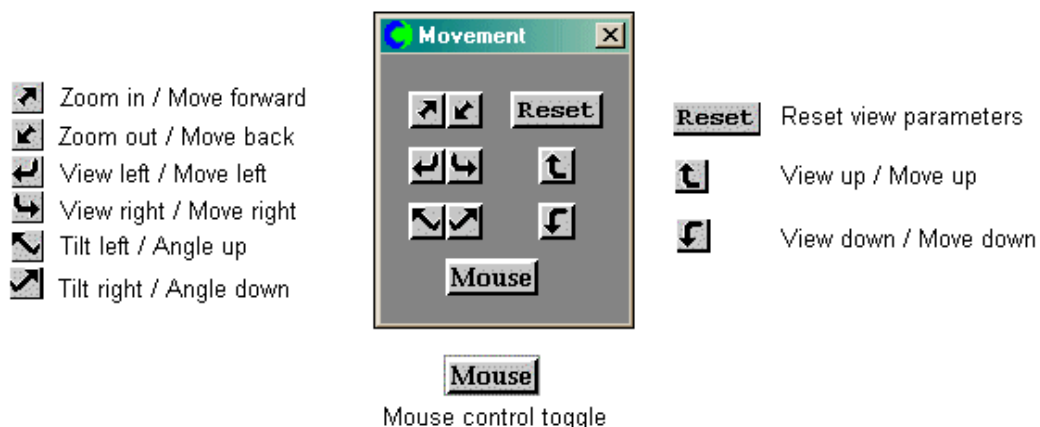
### 1.3 The VR Editor Handset

This is the most useful tool. Some of its functionality is described in the following diagram.












Object and Domain Controls



Movement Controls



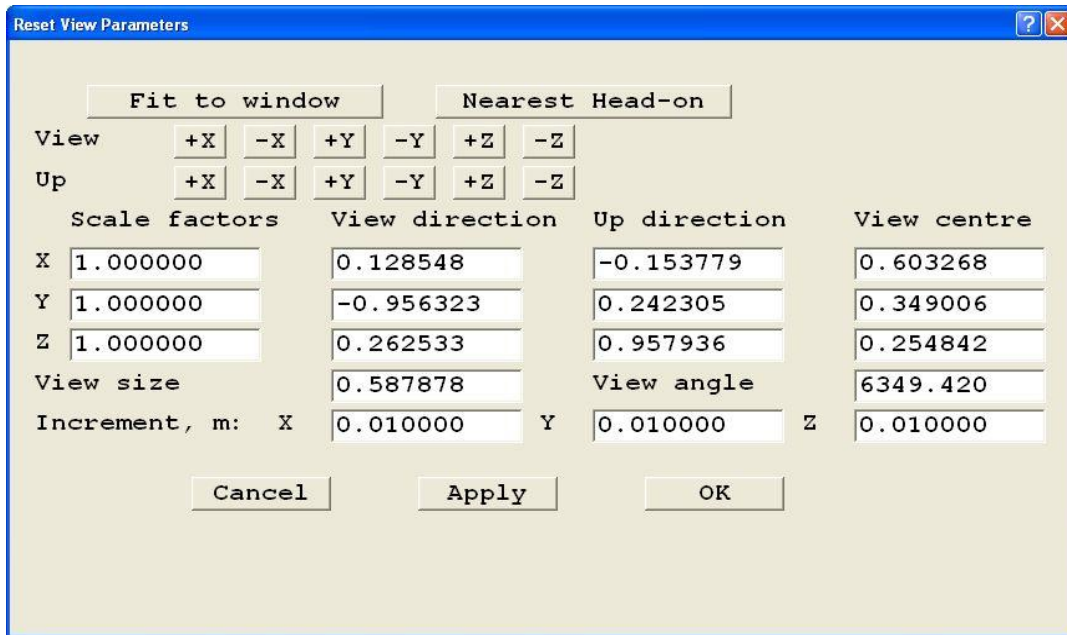
The mouse control toggle ('on' by default) allows the user to move, rotate and zoom the model by using the mouse instead of the arrow keys. The table below explains it in more detail.

Mouse button held down	Mouse movement	Effect	Equivalent key
Left	Left	View left	left-click on 
	Right	View right	left-click on 
	Up	View up	left-click on 
	Down	View down	left-click on 
Right	Left	Tilt right	left-click on 
	Right	Tilt left	left-click on 
	Up	Zoom out	left-click on 
	Down	Zoom in	left-click on 
Middle (or both)	Left	Move left	right-click on 
	Right	Move right	right-click on 
	Up	Move down	right-click on 
	Down	Move up	right-click on 
Shift+Left	Left	Rotate image right about viewer	None
	Right	Rotate image left about viewer	None
	Up	Rotate image down about viewer	None
	Down	Rotate image up about viewer	None
Shift+right	Left	None	None
	Right	None	None
	Up	Move back	right-click on 
	Down	Move forward	right-click on 

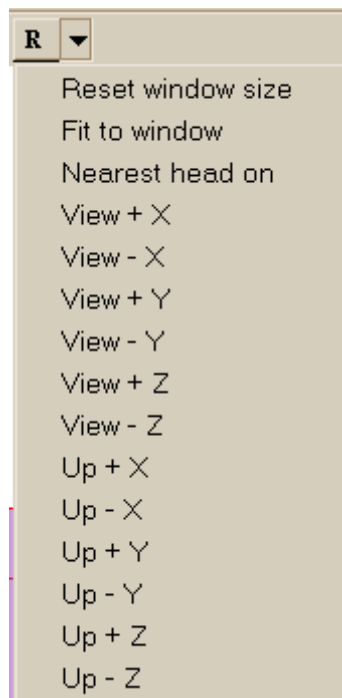
### Quick Zoom

To zoom into a region without panning and dragging, place the cursor where the centre of the zoomed image is to be. Press the control key <ctrl>, and with it held down, hold the left mouse button down and move the cursor to the left or right. A box will follow the movement of the cursor. When the mouse button is released while the control key is down, the area within the box will be expanded to fill the screen.



The **Reset** button will open the dialog below:

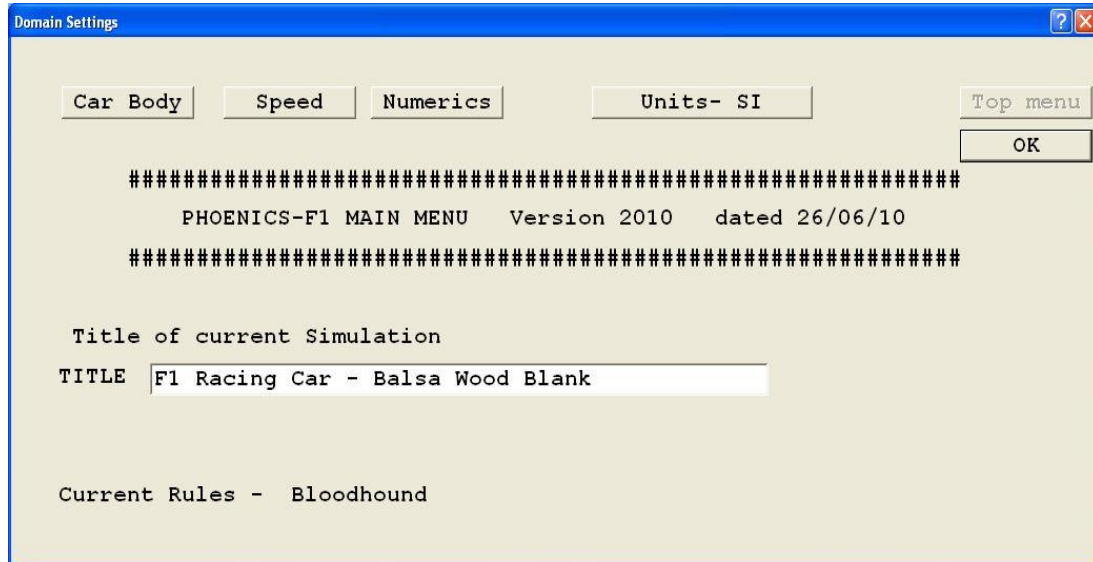


This dialog allows the view direction to be set rapidly. The same functions can be reached more easily from the pull-down list next to the **R** icon on the tool bar:

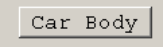
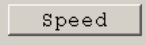

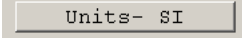


## 1.4 Domain Attributes Menu

Clicking on  in the VR Handset or  on the tool bar will bring up the following window:



There are four options in this menu:

1. Click on  and this will bring up another window concerning the car specifications.
2. By clicking on  the velocity of the car can be altered. By default this is set to 18m/s (the average speed the car is likely to reach).
3. Click on  to bring up another window concerning meshing and solver settings.
4. Click on  and this will switch between S.I. (Systeme Internationale) units and F.P.S. (Foot - Pound - Second) units\*.

The title of the simulation can be changed from '**F1 Racing Car - Balsa Wood Blank**' as shown above, to a suitable title of choice.

\*For the purpose of this document all examples are shown in S.I. units.

1.4.1 **Car Body** window sub-menu:

Domain Settings

Car Body Speed Numerics Units- SI Top menu

Car Body (Included- Wheels No CO2 canister No )

Geometry file balsa

Ride height 0.005000 m Tweak Body Size Go

Wheels Front Back

Diameter 0.035000 m 0.040000 m

Width 0.005000 m 0.020000 m

Axle position 0.050000 m 0.280000 m

Axle length 0.055000 m 0.055000 m

Geometry frnt\_whl back\_whl

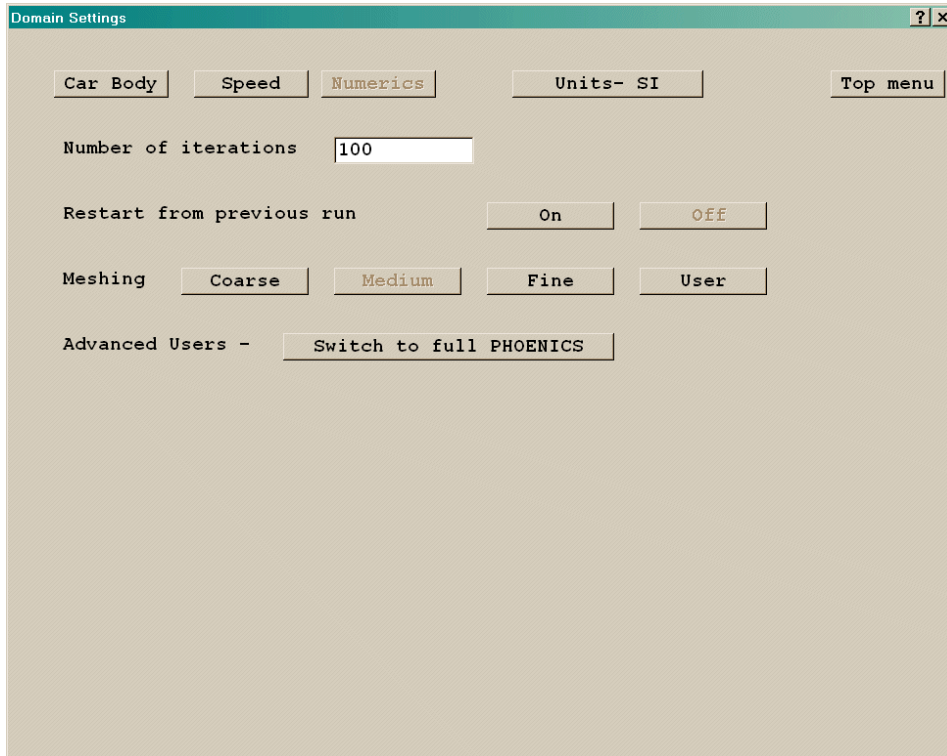
CO2 height 0.024000 m

Legality check On

Clicking on '**Geometry File**' allows the user to select a new model in STL format. This can be generated in CAD, for example using PTC's Pro/Desktop.

- If wheels and/or CO<sub>2</sub> canister are included in the geometry file to be imported, then the user should select '**Yes**' in appropriate '**Included-**' options to the right of 'Car Body'. If not, these options default to '**No**'
- The ride height, wheel diameter and width, CO<sub>2</sub> canister height, axle position and axle length can all be inserted into their appropriate boxes.
- When entering these values it is advisable for the legality check to be switched '**On**' in order to keep the car within competition rules.
- The position of the wheels suits the default balsa wood block geometry. When importing another body, care must be taken to ensure that the axles and wheels are aligned with the new body. Distances are measured from the nose of the car.
- Clicking on '**Go**' alongside the '**Tweak Body Size**' option will allow the user to change the car body dimensions in length, width and height without having to return to the CAD program. Go back to the first menu by clicking on '**Previous Panel**'.

1.4.2 **Numerics** window sub-menu:



In this window the number of iterations can be set (default is 100). Typically 500-1000 may be needed to reach a truly "converged" solution.

- Initially, the restart option should be checked as **'Off'** (as shown above).
- The option is also here to select the grid type. A coarse grid will give the least accurate solution but with the fastest run time, the fine grid will give the reverse. The grid sizes are:

	Cells in X	Cells in Y	Cells in Z	Memory	Time*
<b>Coarse</b>	60	42	30	12Mb	40 Mins
<b>Medium</b>	105	62	60	60Mb	7 Hrs
<b>Fine</b>	145	82	75	140Mb	14 Hrs

\*Using 1 GHz PIII

- The final option on the menu is **'Switch to full PHOENICS'**. This option should not be activated unless in the presence of a teacher, only applied to non-F1 VWT applications, and used in conjunction with the standard PHOENICS User documentation.

1.5. **The Probe**

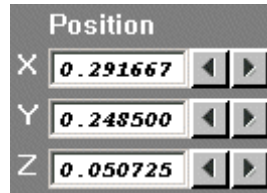



The probe can be placed anywhere within the domain. However, it should be positioned so that during the solution stage it can pick up values for the **velocity**,

**pressure and turbulence terms**, from a useful region around the car. For example, it is no use having the probe on the road in the corner of the domain, but much more sensible to position it behind the car in the flow field.

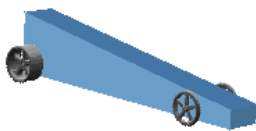
The probe position can be set by:

- Using the arrow keys in the VR Handset can set the probe position:

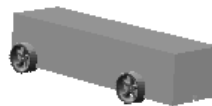


- Typing in the desired location (x,y,z).
- Double-clicking on the probe or on the  probe icon on the tool bar, and using the position dialog that appears.

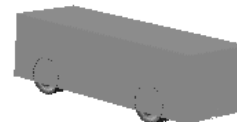
### 1.6. The Original Shape



F1-Bloodhound body



F1-Formula One body



F1-GT body

This is the shape of the physical balsa wood block before any modification; it can be easily replaced with a different geometry using the menu options as described.

- The F1-Bloodhound, F1-FormulaOne or F1-GT balsa wood block, along with the four wheels and CO<sub>2</sub> canister, is the default option within the **VWT**.
- The other three objects in the domain (Road, Inlet and Outlet) are required in the **VWT** for functionality purposes and **should not be changed**.

### 1.7. Car Body Orientation and Size

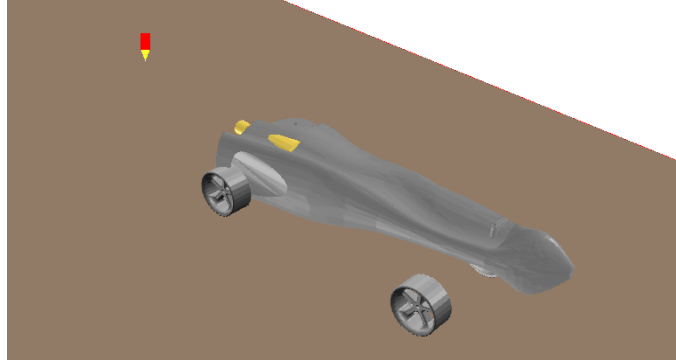
A very important point is the orientation of the co-ordinate system used in the STL file. F1 VWT is set up with the car pointing in the +X direction, the car width in the +Y direction, and +Z pointing up.

The F1 STL import software will assume that:

- The largest dimension is along the length of the car body.
- For Formula-1 and GT that the car width is greater than the height, for Bloodhound that the body height is greater than the width.
- The units are assumed to be millimetres.



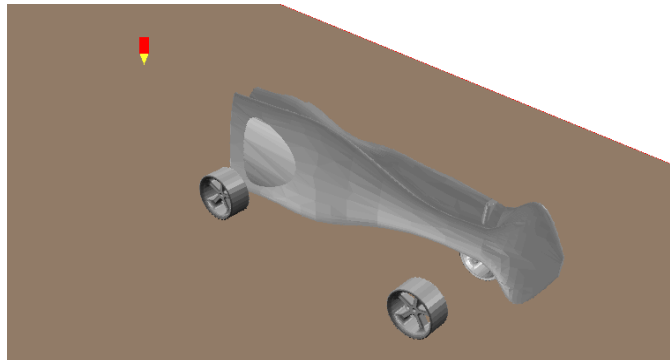
If the body STL has been generated with these constraints, it should be imported correctly. If the STL has been generated in another orientation, it is usually still possible to rotate the body to face the right way.




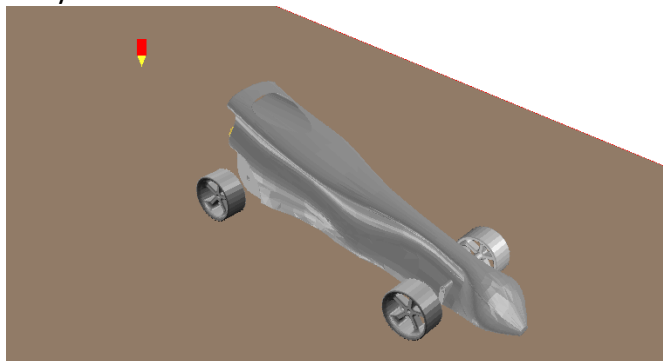
In this example, the body is taller than it is wide, and so has been imported lying on its side. This needs to be rectified by swapping the height and width, then rotating the body into the correct orientation.

In the  panel turn the legality check off, then click 'Go' next to 'Tweak Body Size'. On the new dialog, you should see that the Length, Width, and Height values are mixed up. Enter the correct value into each box. Click 'Previous panel', turn the legality check back on then 'OK' to close the Main Menu.

The body should now have the correct dimensions, but will still be orientated wrongly.




Click on the body to select it. It will be highlighted to confirm the selection. Now click on the Rotate Object Up  button on the Editor handset (second icon from right, top row). Each click will change the orientation of the body within its box. Continue clicking until the body orientation is correct.



NB. If the units in the imported STL file are not millimetres, the body size will be wrong. In that event, go to the 'Tweak Body Size' page described above and enter the correct sizes (in metres or feet depending on the chosen unit system.) Cancel any warnings about illegal dimensions.

### 1.8 Getting help

To get help on a menu button or data entry box, click help on the  in the top right-hand corner of the dialog window, then point to the item in question and click again. The tool bar and handset icons have a bubble-help which appears if the cursor is held steadily over the icon for more than 1 second.

## 2. The Solver – Earth

### 2.1 Overview

Once the setup of the simulation has been completed, the next stage is to run the analysis. This is achieved by clicking on **'Run'** on the Menu Bar using the following procedure.

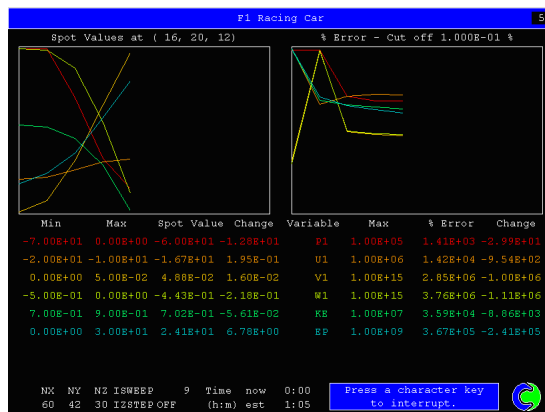
**Menu Bar:**

**Run...**

⇒ **Solver**

This opens the solver screen where you can monitor your simulation as it moves towards a solution:

Window at commencement of run



Window below shows nicely converging solution - 500 to 1000 iterations (sweeps)



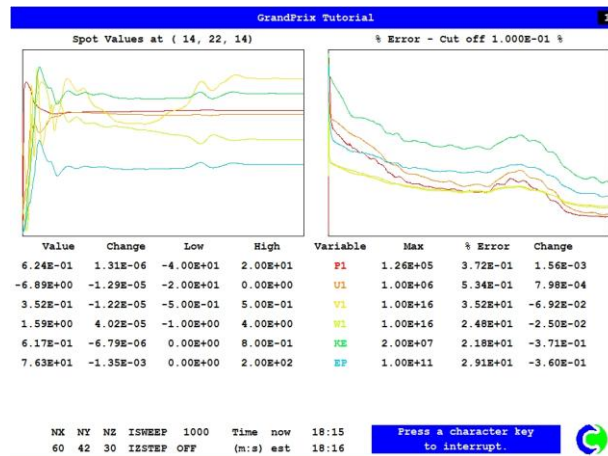
- The solver is working out a solution for **pressure (P1)**, **velocities (U1, V1, W1)** and **turbulent quantities (KE and EP)**. U1, V1 and W1 are the velocities in the X, Y and Z directions respectively. KE is the turbulent kinetic energy, and EP is the rate of dissipation of KE.
- In each image, the graph on the left shows the **'Spot values'**, at the probe position in the VR Editor. If the U1 velocity looks sensible, then your simulation is probably setup okay. The lines on this graph should eventually become horizontal, showing that the values have stopped changing.
- The graph on the right shows the sums of errors for each equation, which indicates how close the solver is to a final answer. This is recognized when the lines for each variable decline and eventually approach the bottom axis. If all the lines reach the bottom axis, the solver will stop. **This situation in any simulation is known as convergence.**
- The axes on the graphs represent the numerical change in each variable (along the vertical axis) and the number of iterations (along the horizontal axis). This indicates the importance of the iterations set in the editor; if there aren't enough, the simulation may never reach convergence and hence a final answer.

## 2.2 Assessing Convergence

The equations for velocity, pressure and turbulence quantities are solved in a guess-and-correct manner. Details of this procedure can be found on-line at <http://www.cham.co.uk/ChmSupport/intrlecs.php>.

On each iteration, or sweep, the values change a little. When the change from sweep-to-sweep becomes acceptably small, we have **convergence**. If the changes keep getting bigger until eventually the solver blows up, we have **divergence**. The default settings in F1 should result in convergence.

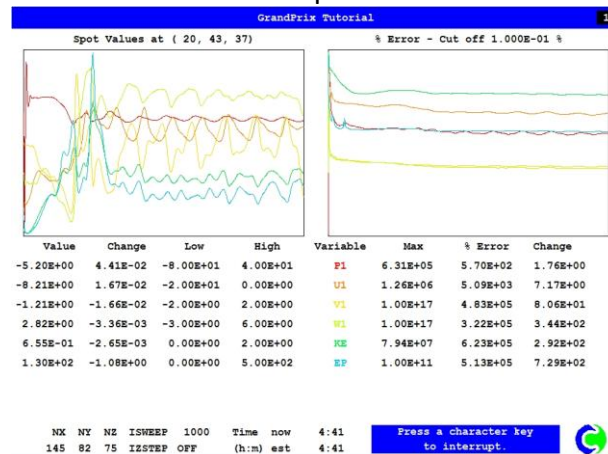
A monitor plot like the one shown on the right above, or here indicate good convergence.



If the plots show the right trends, the right-hand graphs generally heading downwards, and the left-hand graphs beginning to flatten out, but the left-hand values are still changing appreciably you will need to do more iterations.

To do this, on the **Numerics** sub-menu click 'Restart from previous run' to ON then run the solver again. Repeat the process several times, perhaps running 500 iterations per run. Three or four such continuation runs add up to 1500-2000 iterations which should be enough.

There can be occasions when the monitor plot looks more like this:



The errors (on the right) are much bigger, and the values at the probe location (on the left) are oscillating about a fairly steady value. This kind of plot almost certainly indicates that the model is trying to predict **vortex shedding** behind the car, but cannot do so because it is running steady-state. This is more likely to happen with medium or Fine grids, as with the Coarse grid there may not be enough cells to capture the vortices properly.

As the vortices are being shed behind the car, their effect on the drag and lift is minimal, and the values obtained can be used.

For more general information on vortex shedding, see [http://en.wikipedia.org/wiki/Vortex\\_shedding](http://en.wikipedia.org/wiki/Vortex_shedding).

The general PHOENICS code can predict this when run in time-dependent, transient mode, but this is outside the scope of the F1 specification.

### 3. The Post Processor - VR Viewer

The post processor enables the study of the results from the simulation. The Viewer can display **vectors**, **contours**, **streamlines**, **iso-surfaces** and **surface contour plots** of the variables solved for. Examples of how to generate some of these are shown in the tutorial.

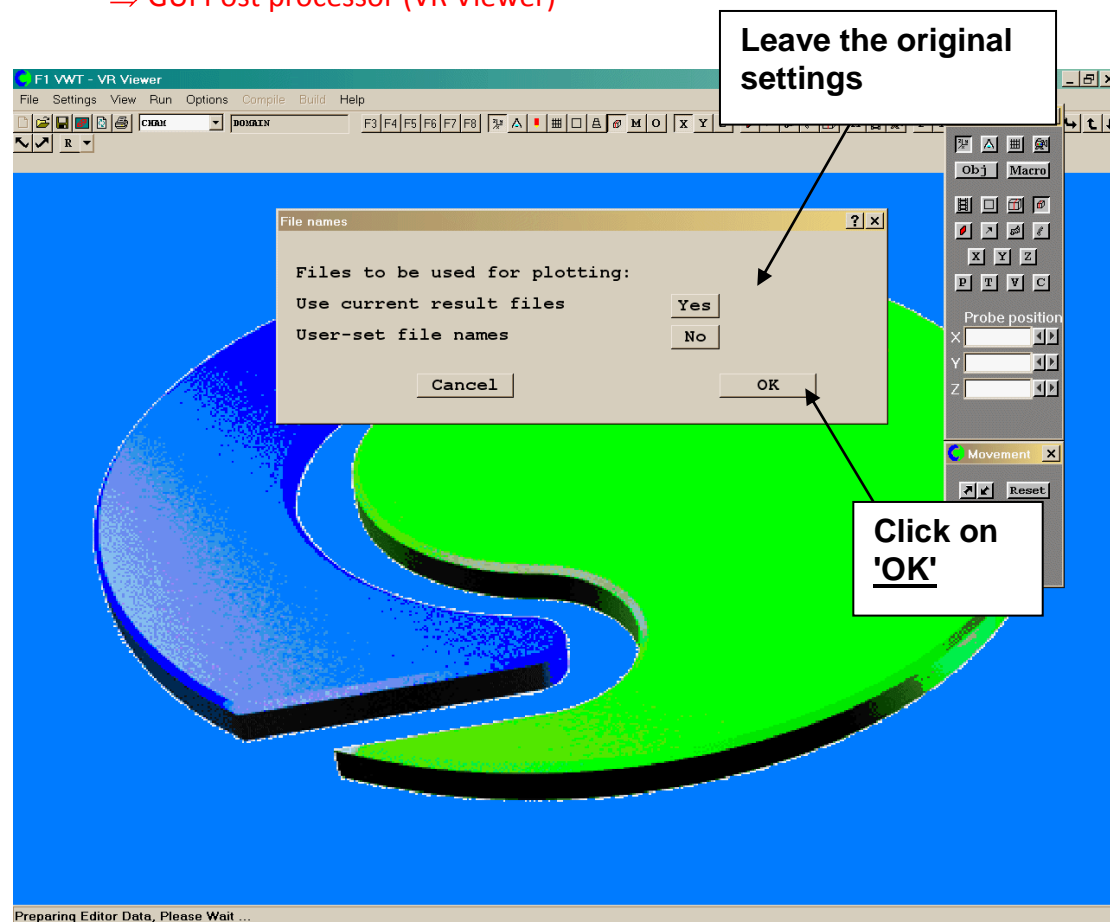
To load up the viewer click on '**Run**' and select '**Post processor**' then '**GUI – Post processor (VR Viewer)**', This will open up a box '**File Names**', keep the original settings and click '**OK**' to open the viewer.

**Menu Bar:**

**Run...**

⇒ Post processor

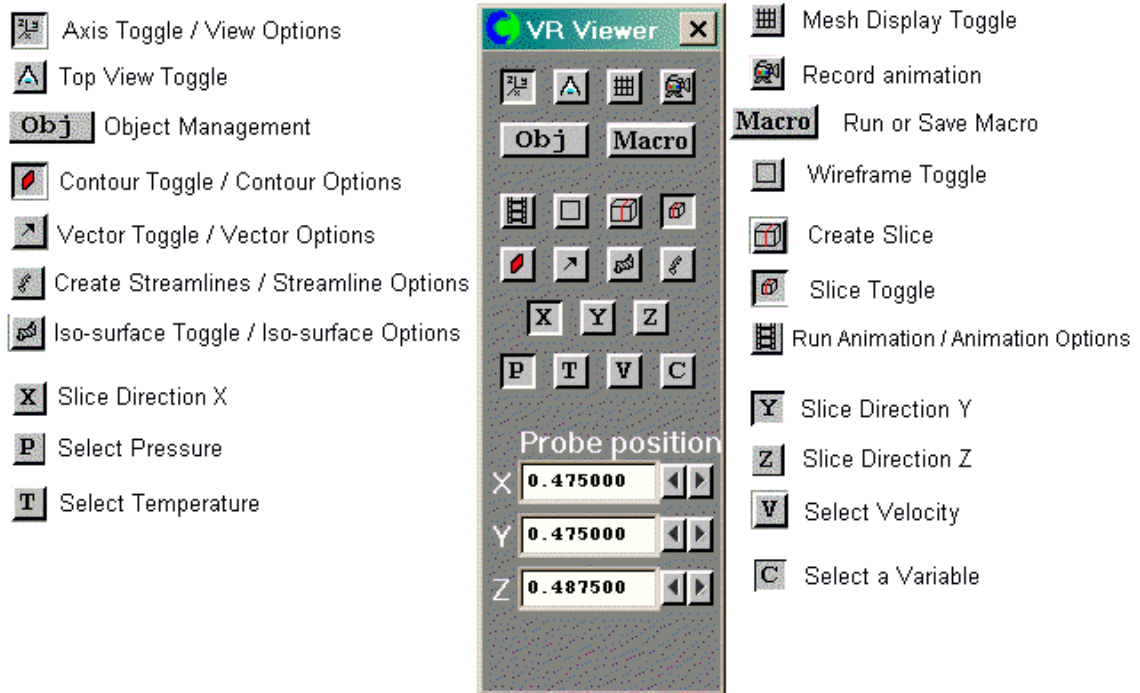
⇒ GUI Post processor (VR Viewer)



The viewer is similar in format to the editor; the important part is again the handset, shown on the next page.

### 3.1. The Post Processor – VR Viewer Handset

#### Viewer Controls



The movement controls are as for the Editor, described earlier.

The same icons also appear on the Viewer tool bar.

This concludes the overview of the VWT. To learn more about the functionality of the program please study the demonstrations cases and complete the tutorial.

## Virtual Wind Tunnel – Tutorial – Bloodhound Class

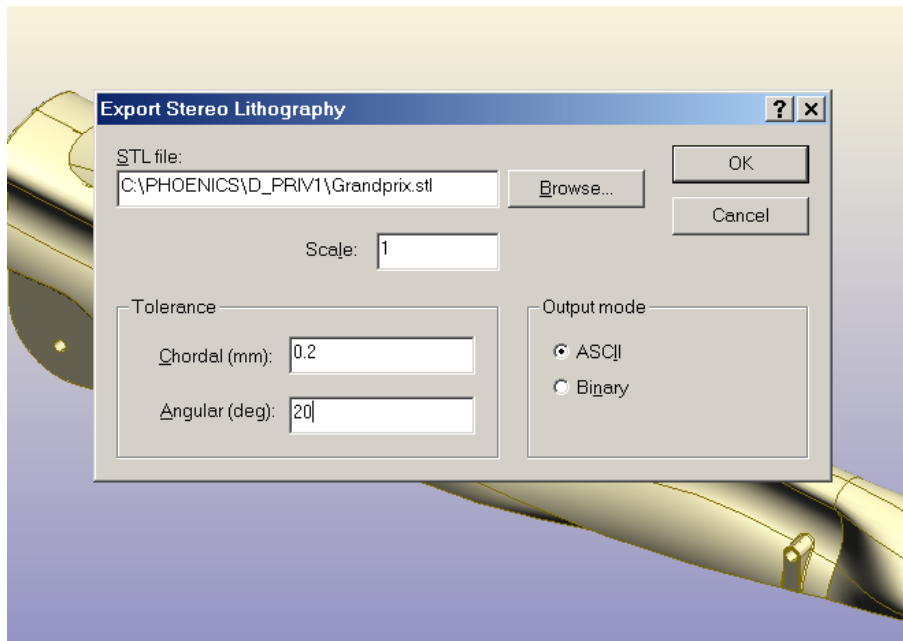
This tutorial follows on from PTC's 'Grand Prix 2000ii 2D' Pro/Desktop tutorial. Within this session the functionality of the Virtual Wind Tunnel (VWT) will be demonstrated from exporting suitable car geometries from Pro/Desktop, through obtaining a solution, to viewing the results generated. It is recommended that the user read 'VWT Overview' before starting this tutorial.

### 1. Generation of Stereo-Lithography (STL) files.

In the Pro/Desktop environment, click on 'File' then 'Export' and select the 'Stereo-Lithography' (STL) option.

**Note:** Use the completed geometry from 'Grand Prix 2000ii 2D'.

This will bring up the window:



Make sure that the STL file is exported into the PHOENICS working directory, as shown in the path line. **[Drive letter]:\PHOENICS\D\_PRIV1\[Filename.stl]**.

Type '20' in the Angular (deg) box.

**Note:** This is important as values much lower than this create large files that may prove difficult to read into the VWT.

Click 'OK' and Grandprix.stl will be exported into the working directory.

The final and important point is the orientation of the co-ordinate system. The F1 STL import software will assume that the largest dimension is along the length of the car body, and that the body height is greater than the width. The units are assumed to be millimetres.

If you have trouble with generating this body shape, a copy of the **grandprix.stl** file can be found in **\phoenics\d\_intfac\d\_cadpho\d\_stl**. Copy this file to **\phoenics\d\_priv1**, and proceed with the tutorial.

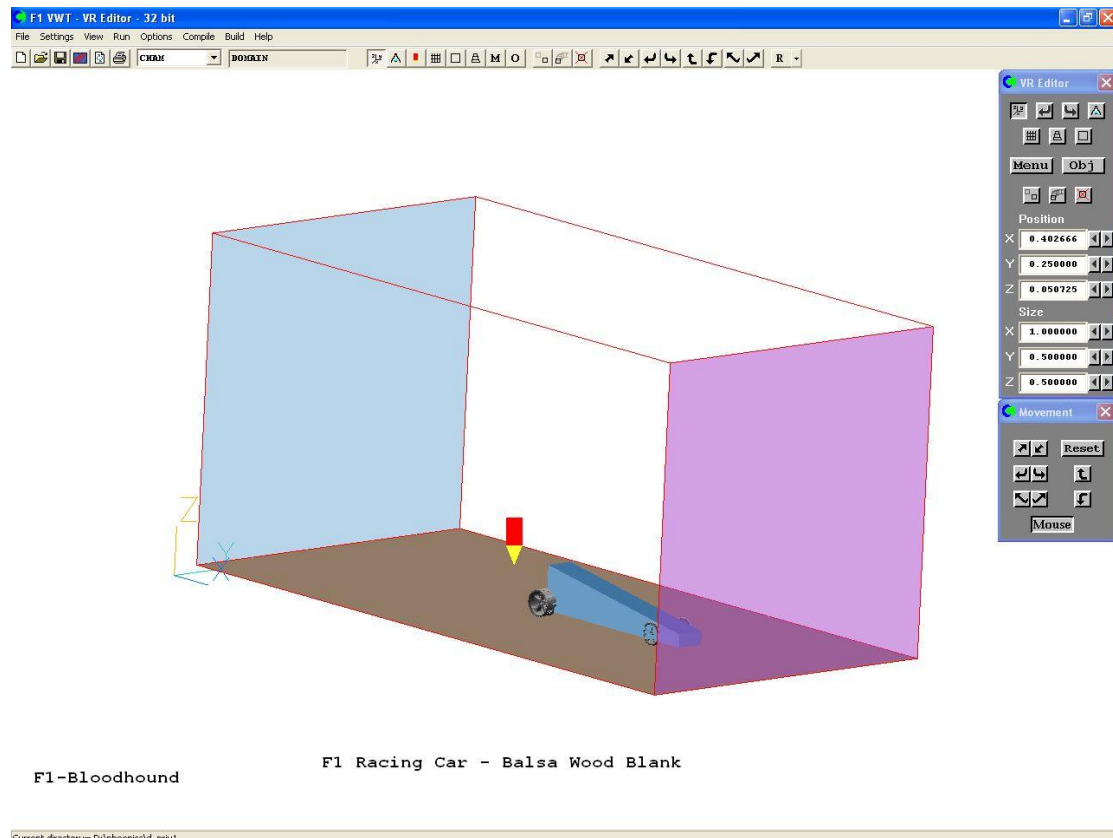


## 2. Starting with VWT

Click on the VWT icon ...



... which loads up the Virtual Reality (VR) Editor - the environment where all the setup of the VWT takes place:



Notice, to start with the default geometry of the car (wedge shape) is available for analysis. The four wheels and CO<sub>2</sub> canister are all standard in the VWT domain.

To select a Bloodhound class blank body, click on **File, Start new case**, and select **F1-Bloodhound** from the list. For a Formula One class blank, select **F1-FormulaOne**.

The images for this tutorial have been taken from a Bloodhound class case. The basic procedure is exactly the same for a FormulaOne class.

- Click on **'Menu'** located on the VR headset, and change the title of the simulation to 'GrandPrix Tutorial'.

### 3. Importing the geometry

- Click on  and select the '**geometry file**' option (currently selected as '**balsa**'). This will open up a dialog in which the STL geometry file should be selected.

From the '**Car Body**' window, the position and the size of the wheels can be altered. Check that the following is true and amend as necessary:

	Front (m)	Back (m)
Wheel diameter	0.035	0.04
Wheel width	0.005	0.02
Axle position	0.050	0.28
Axle length	0.055	0.055
Ride Height (m)	0.005	
CO <sub>2</sub> canister height (m)	0.024	

**Note:** To keep the car within specified competition rules the legality check should be clicked '**on**' (default option).

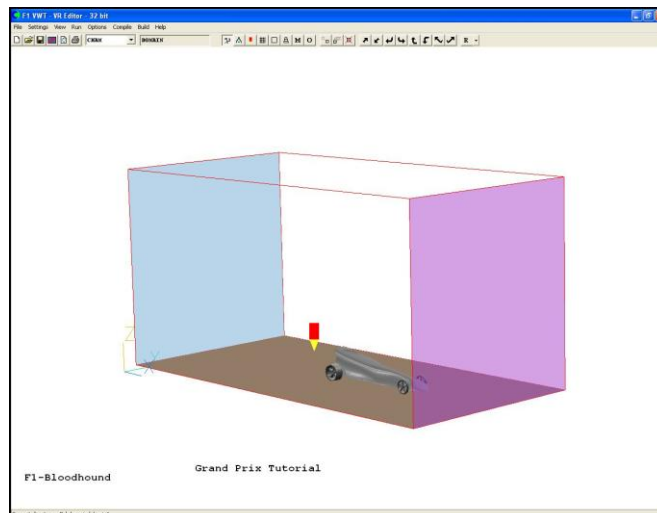
- Now click on  (located next to '**Tweak Body Size**').
- Check that the length, width and height are as expected (as generated by Pro/Desktop). In this case they should be **0.29 m**, **0.042 m** and **0.068 m** respectively.

#### **REMINDER:**

**FPS units can be selected for data inputs instead of SI units at any time.**


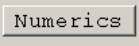
- Click on  and then on '**OK**'; this returns you to the main **VR environment**.

Notice that now the wedge shaped blank has been replaced by the Grand Prix geometry (below).

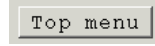



#### 4. Setting up the VWT solver (Earth)

The next stage is setting up to simulate the airflow around the Grand Prix car.

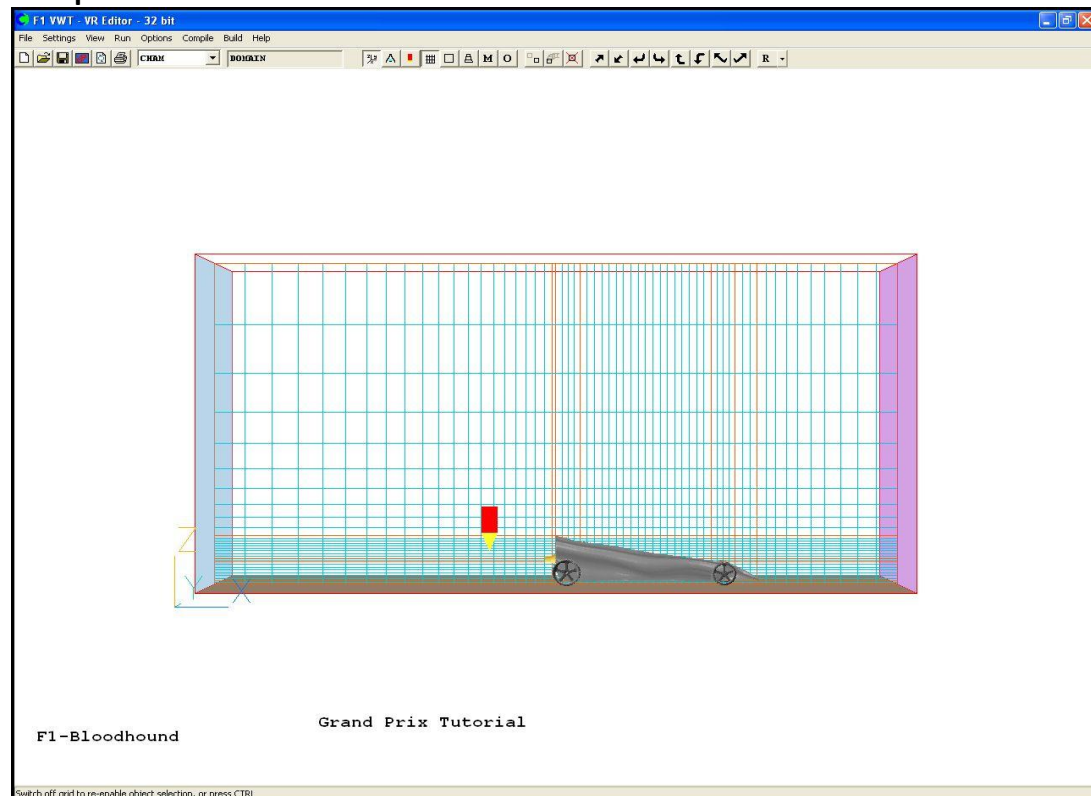
- Click on  and select the 'speed' option, here the '**Running Speed**' is set but for the purposes of this tutorial it should be left at **18 m/s** (default).
- Click on . This window gives you the option to select the grid (coarse, medium, fine or user) and the number of iterations. For this tutorial the following selections should be made:
  - Number of Iterations = 300 (default = 100)
  - Grid = Coarse.

**NOTE:** It is important that the '**Restart from Previous Run**' option is set to '**off**' (default). This window also gives the user an option to switch to full PHOENICS, but this should only be attempted in the presence of a teacher.

- Click on  and then '**OK**'. This will take you back to the main VR environment.
- The grid specified in the previous window can be viewed in the VR Editor by clicking on the Mesh Toggle  in the VR Handset.

By rotating the view, the grid can be displayed on each of the coordinate axes.

#### Example



Notice that even in this coarse grid example, the grid is significantly finer in the areas immediately around the car. This is because, as CFD analysts, we are only interested in the flow around the car and not, for example, at the top of the domain where the car aerodynamics are not affected.

This completes the setup of the simulation.

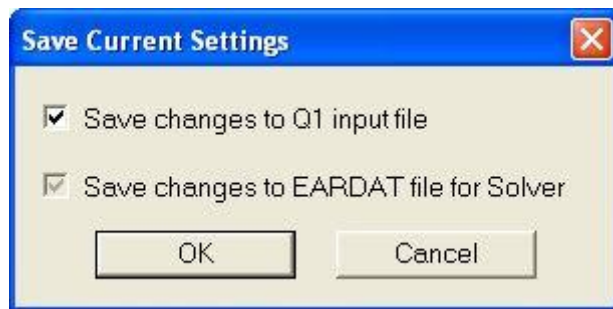
### 5. Running the VWT's solver (Earth)

In the **VR Bar Menu** - main title bar, follow the procedure:

Run...

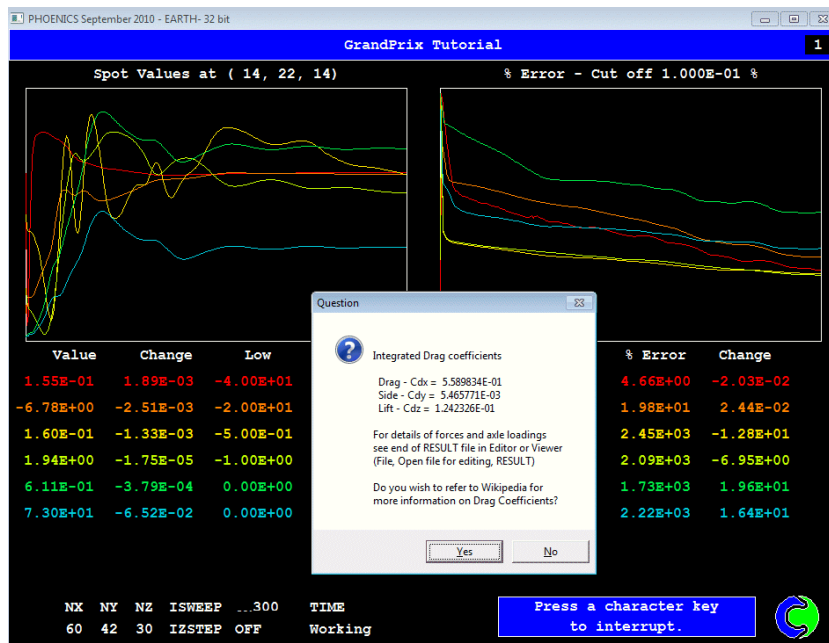
⇒ Solver

This will generate a box like this;



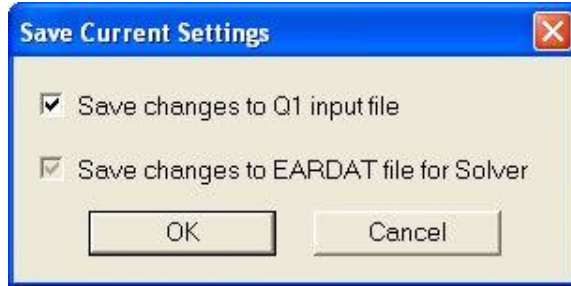
- Click '**OK**' and the solver will start.

**Earth** will solve for 300 iterations, this will be enough for the simulation to converge. The run time for this example is approximately 20 minutes on a 3GHz Pentium 4 computer, giving a convergence graph looking like this:



At the end of the run a dialog will appear showing the total forces and drag coefficients on the car in each of the three directions, and indicating how the full drag results can be seen. Further details on this are given in Section 7. An option is offered to open the Wikipedia article on Drag Coefficients ([http://en.wikipedia.org/wiki/Drag\\_coefficient](http://en.wikipedia.org/wiki/Drag_coefficient)).

Now the simulation has finished the next step is to save the results. This is done by clicking '**File**' in the VR Bar Menu and select '**Save as a Case**'. When the following dialogue box comes up, click **OK**:



Now type '**Grandprix**' in the filename box and click '**Save**'. A box will appear '**Save settings to Q1?**'; click '**Yes**' and the results will be saved into the working directory. Saved cases can be picked up later by clicking '**File**' '**Open existing case**'.

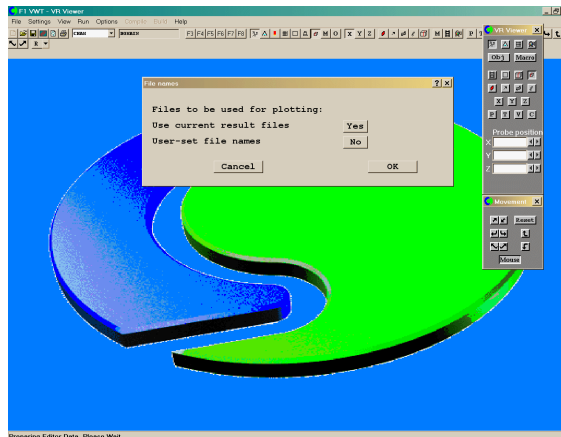
## 6. Viewing the VWT Results

In the **VR Bar Menu** - main title bar, follow the procedure:

**Run...**

⇒ **Post processor**

⇒ **GUI Post processor (VR Viewer)**



- Click '**OK**' and the results will be loaded into the viewer.

The following sections will show the VR Viewer tools applied to this simulation.




**NOTE:** If at any time during the post-processing a screen plot is required, click '**File**' select '**Save Window As**', type in a filename and select either PCX, BMP, GIF or JPG

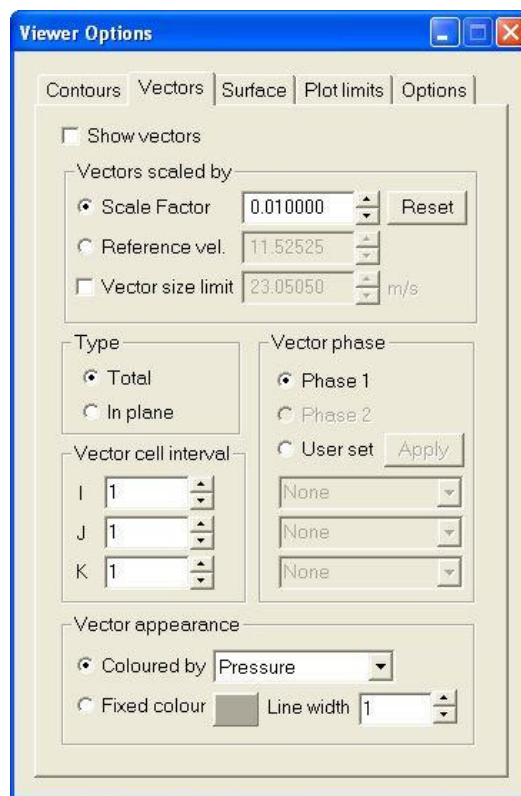
as the extension. Click **'Save'** and everything present in the **VWT window** shall be saved.

The results of the simulation can be visualized as:



- Vectors (arrows showing local flow speed and direction).
- Contour maps showing distributions of pressure or velocity at selected planes.
- Iso-surfaces showing regions of constant value.
- Streamlines showing the path of dust particles dropped at chosen locations.

### 6.1 Velocity vectors

- Click on  to select velocity as the plotting variable and left click on the Vector Toggle  in the VR Viewer handset to display the vector field.
- If you right click on  a window like this appears

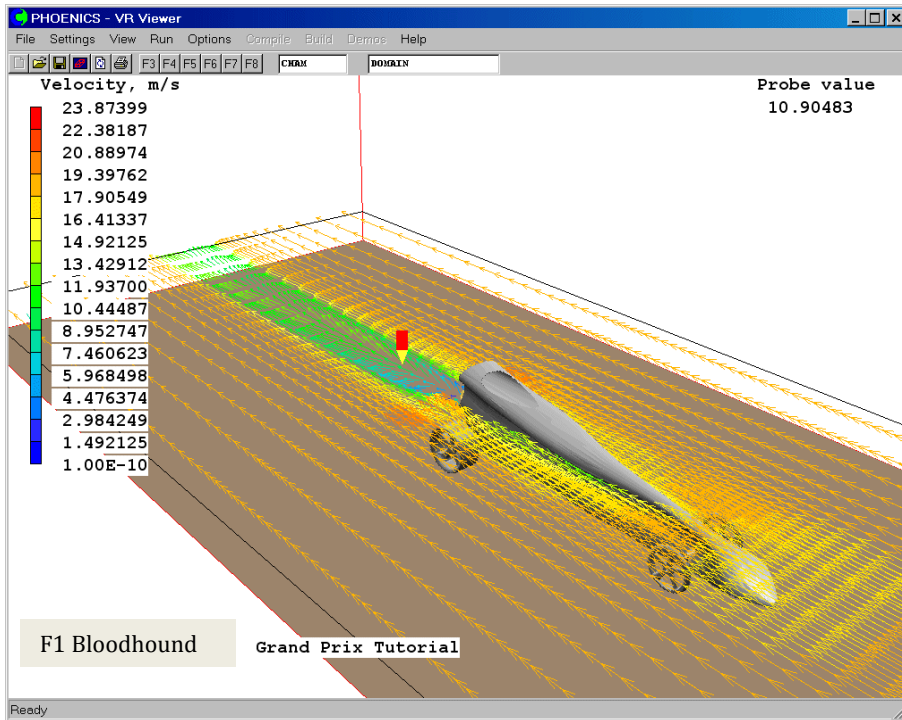


You can then make the vectors shorter or longer if you wish by altering the scale factor.

- Now select the **Z-plane**  and move the probe in the Z direction  to 0.04. The vector plane will follow the probe.

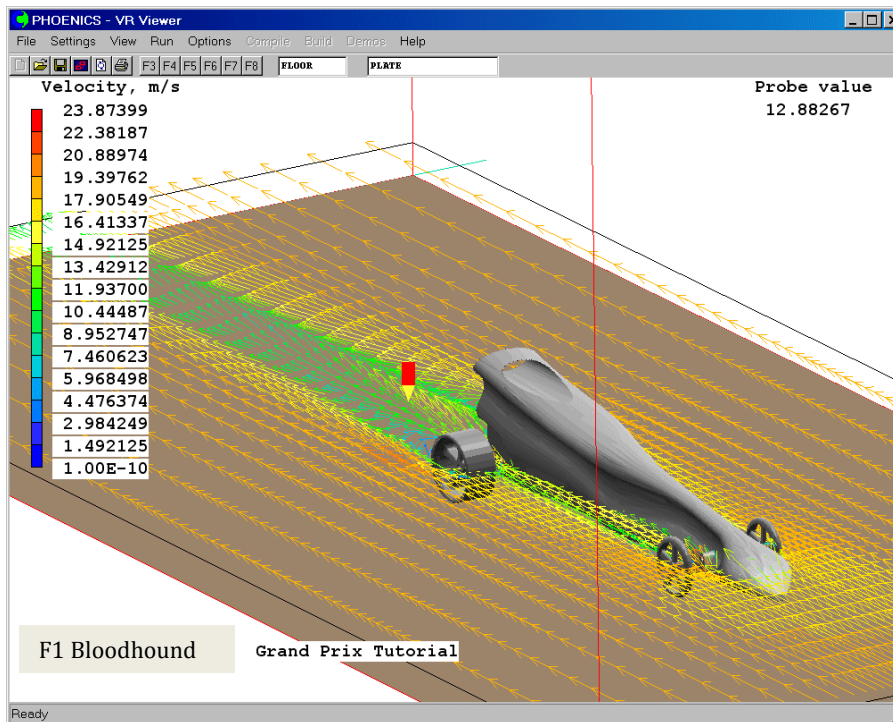
- Rotate the domain until the most appropriate view is attained.  
In the example below, we can clearly see the **wake** behind the body of the car.

**Velocity field - Z plane**

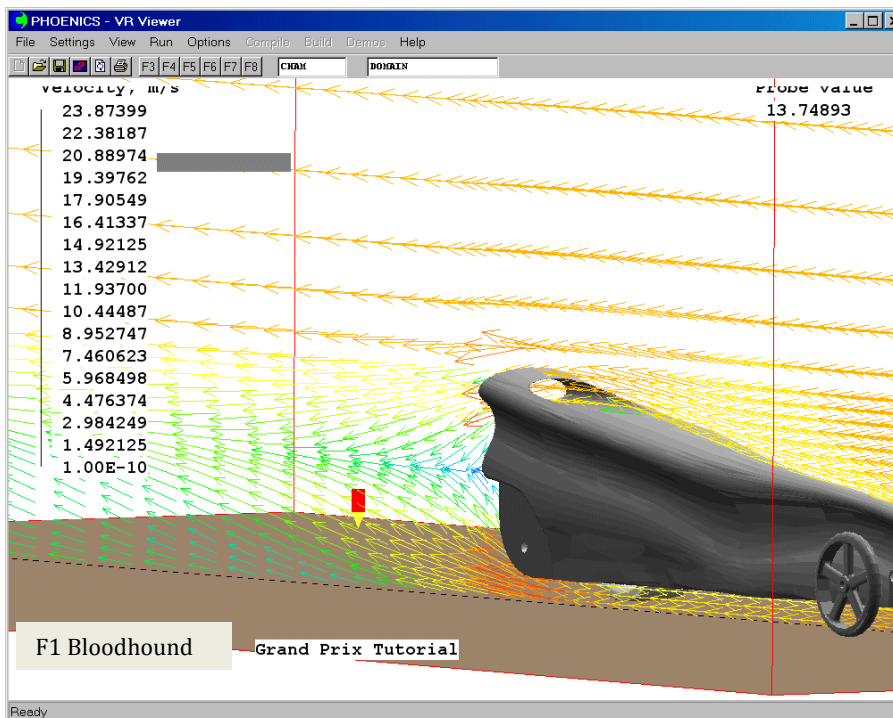


There are many other useful views in all three planes. They are viewed using the same technique described. Some example views are displayed below. (The gap between the car body and wheels has been exaggerated for viewing purposes.) See if you can get similar plots.

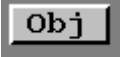
This shows again the Z-plane at a level lower down the car ( $Z=0.02$ ). The velocity can be seen to increase around the wheels, particularly around the wider rear wheels.



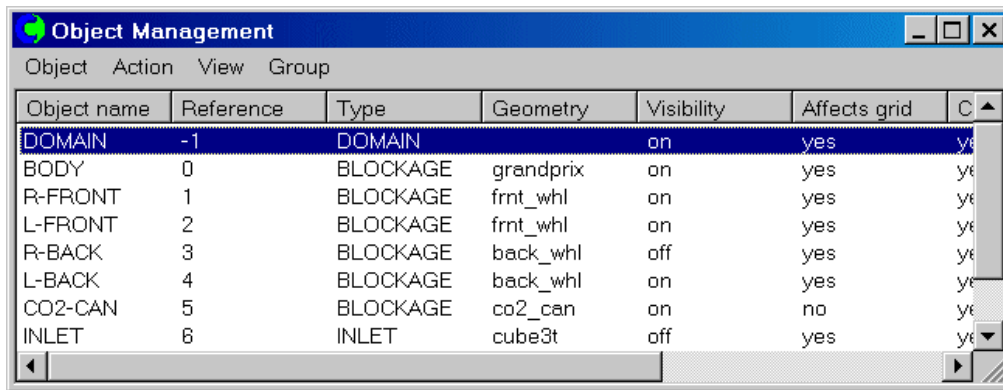
The mid-plane is shown in the Y-direction. The **wake** is again shown behind the car body. The velocity can also be seen to increase underneath the car.



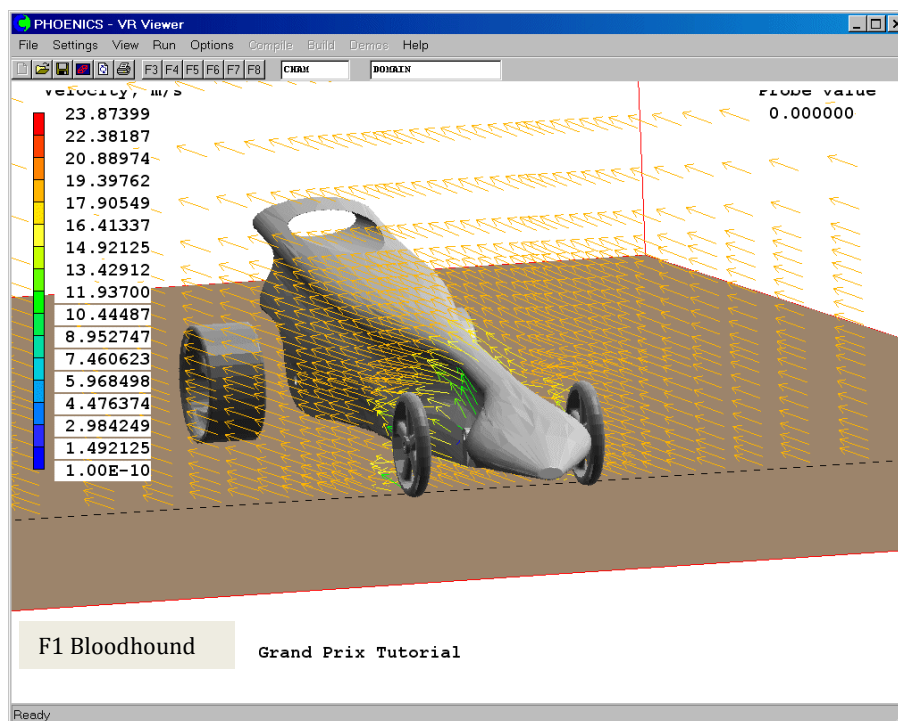
## 6.2 Hiding Objects

Note that the rear wheel has been hidden to get a clearer view of the vectors. To hide an object, left-click on it to select it, right-click to bring up a context menu, and select 'Hide object'. To unhide the object, click on the  button on the hand-set. This will bring up the Object Management Panel, shown below:








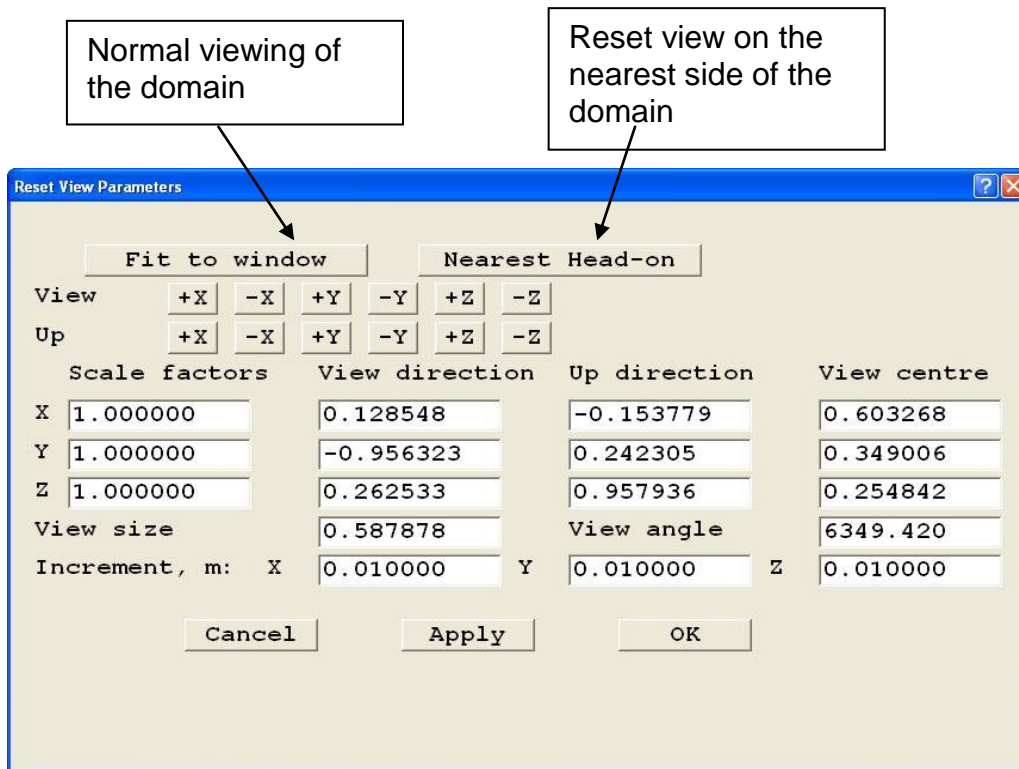
The 'Visibility' flag for the R-BACK object (right-back wheel) is 'off'. Right-click on the R-BACK line in the panel and select 'Reveal object' from the context menu. This dialog can be used to hide or reveal several objects at once, selected using standard Windows selection techniques.



A vector plot taken in the X-plane around the nose of the car, showing detail of the flow near the front wheels.

You can follow the same procedure and display different views by using  plane and different probe positions.

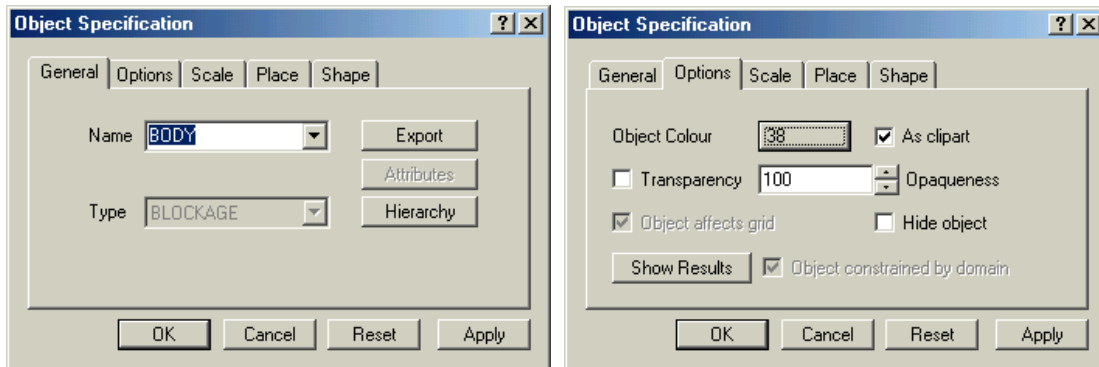
- Use the mouse toggle  for moving the domain orientation
- Use  to set a specific view.



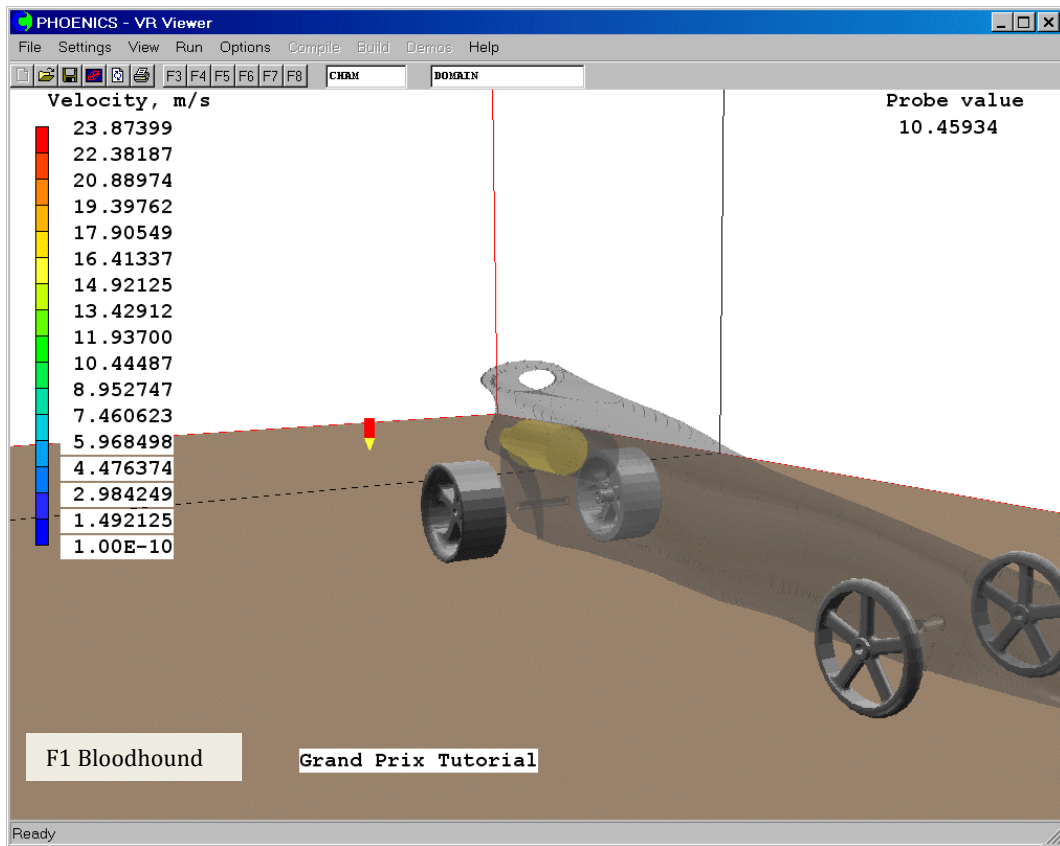
### 6.3 Transparency

The model can also be made to appear transparent; for example, in order to more clearly visualize the flow behind the object. To do this:

Double click on the car body and a window like this will appear






Click on **options**, and then click transparency. The opacity can be altered as you require. 50% is usually a good value. Then click **OK**. To get the transparent image of the car click somewhere else on the screen so the car body isn't highlighted. The image is now transparent.



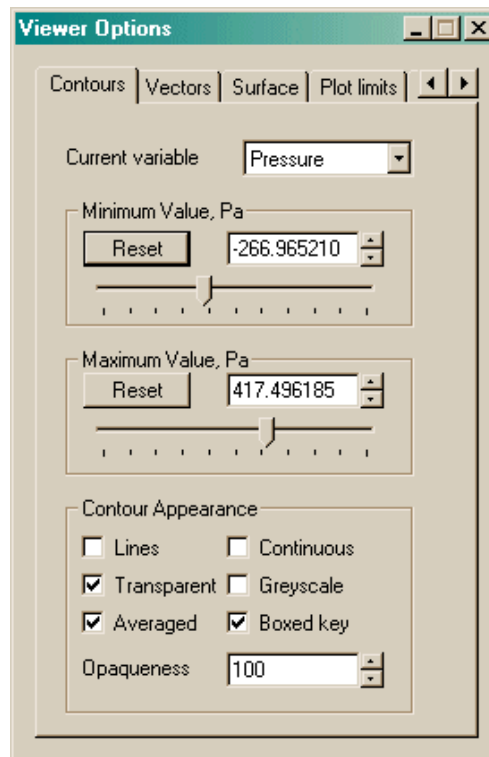
## 6.4 Velocity Contours

These are created in a similar way to the vector plots.

- Click on  to select velocity and then click on the contour toggle  in the VR Viewer Handset or the tool bar.

**NOTE:** If the vector toggle  from the previous instruction is not switched off, both will be displayed simultaneously.

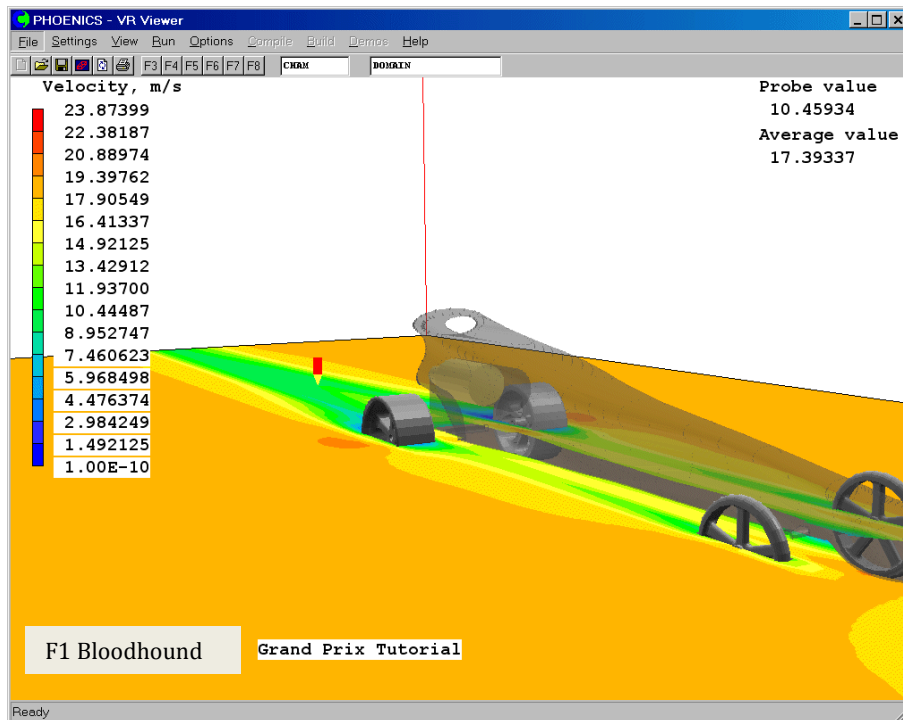
- If you right click on  a window like this appears



You can then alter the contour appearance if you wish.


- Now move the domain orientation to that of the first vector plot (Z-plane,  $Z=0.02$ , using a suitable rotation).

The contour can be made transparent by setting the opacity to say 50%.

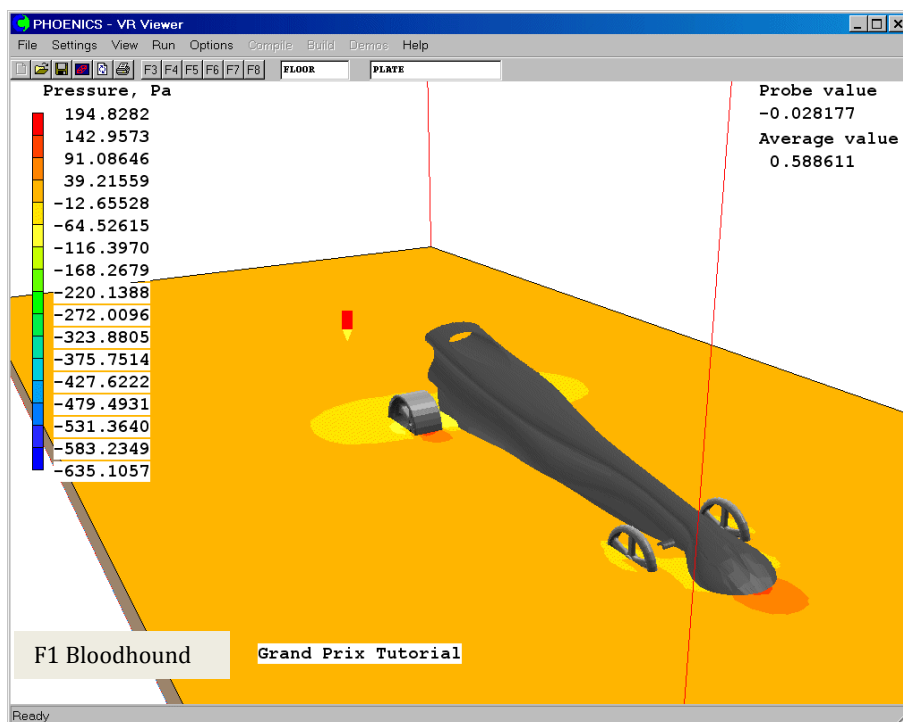


This shows the contoured comparison to the first vector plot. The **wake** behind the car can be clearly seen.



### 6.5 Pressure Contours

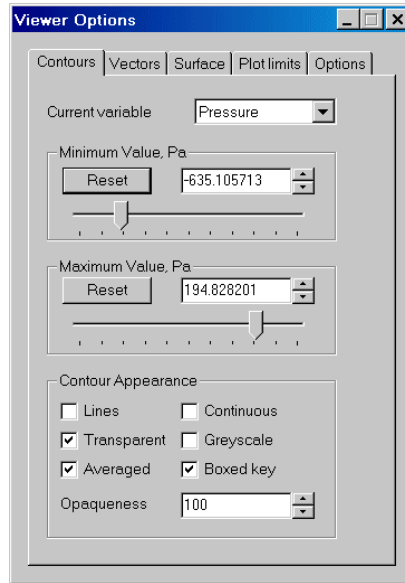
Click on  to select pressure and then select the contour option in the **VR Viewer handset** or tool bar. Move the **Z-plane** to a value of  $Z = 0.02$ .

This gives a pressure plot similar to this.



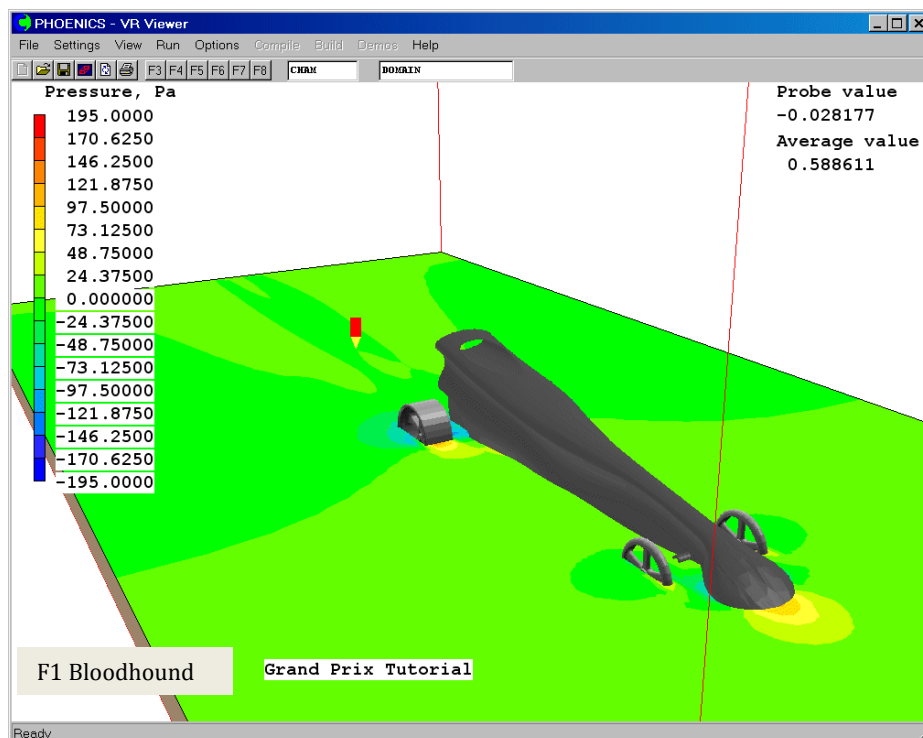
An important point to remember when viewing these plots is to select the appropriate contour or vector scale.

- To set a range for the contours, click on 'C' , or right click 



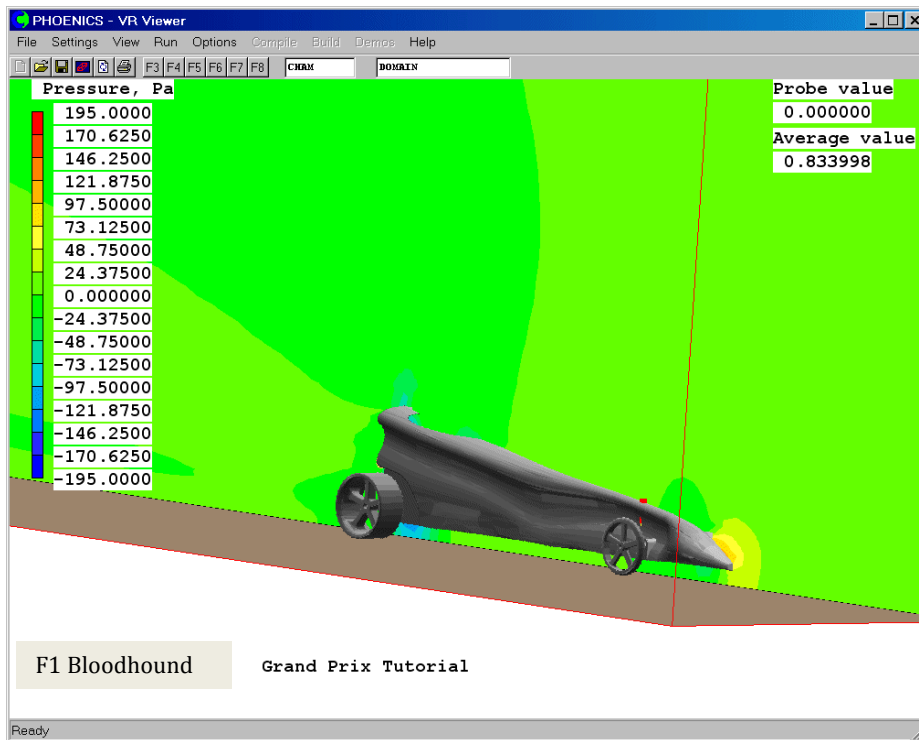
- Change the **minimum value to -195** and **maximum value to 195**, and close the window.



The plot below shows the same information but scaled over a reduced range. This shows up the smaller values better, and can be useful if the largest values dominate the picture and swamp out smaller variations.



Again similar plots can be obtained in the important areas, see if you can produce the one below.

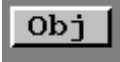
Pressure plot along the mid-plane of the car in the Y direction:



If FPS units have been selected, a second pressure variable is available for plotting. This is called PSIG, and it represents the gauge pressure in pounds/square inch. It can be shown by clicking on , or right clicking , then selecting it from the list of variables next to 'Current variable'. The usual 'Pressure' variable contains the pressure in pounds/square foot.

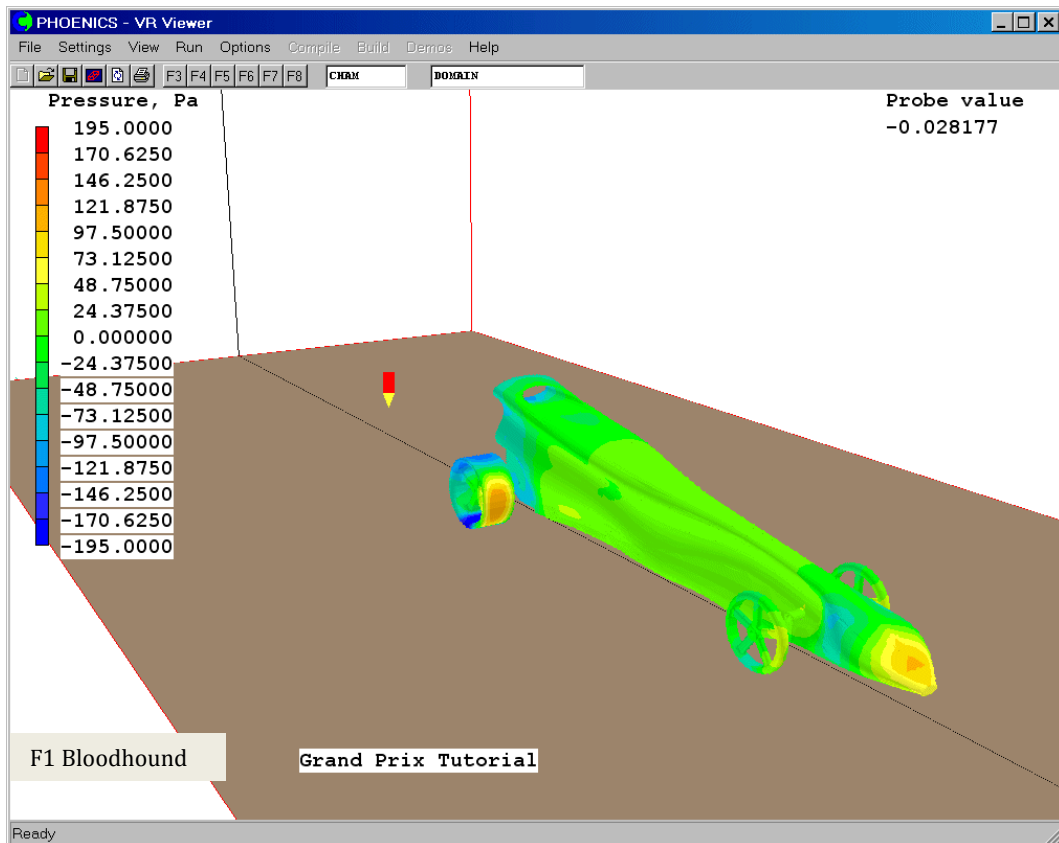
## 6.6 Surface Contours

This tool allows data to be displayed on the surfaces of the geometry.

Click on  to open the object management display.

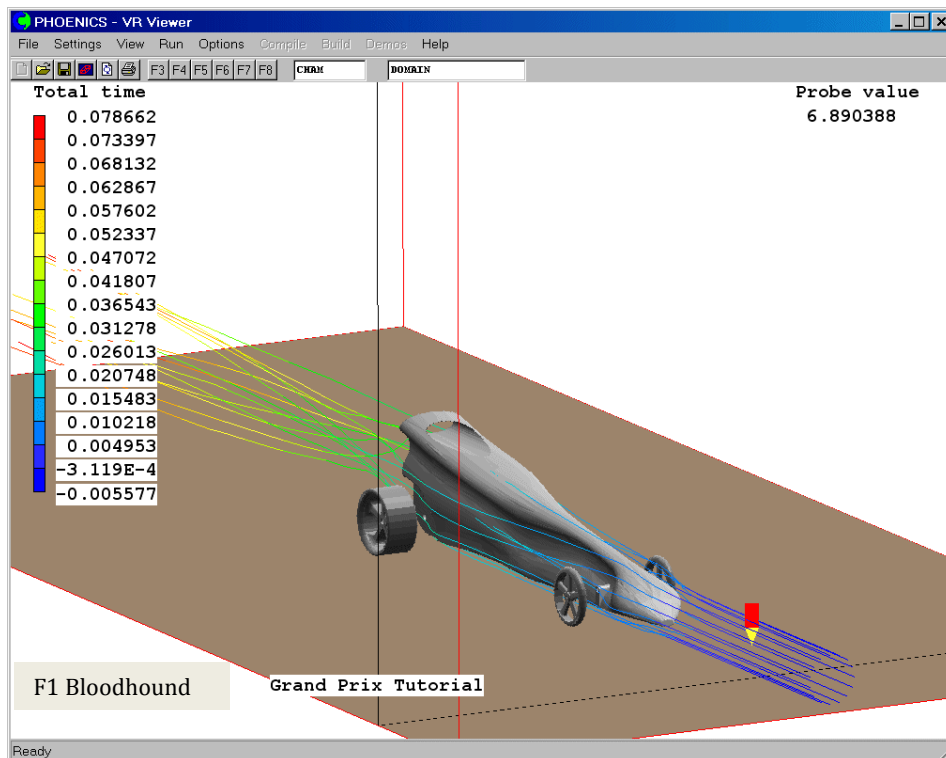
Object name	Reference	Type	Geometry	Visibility	Affects grid	C
DOMAIN	-1	DOMAIN		on	yes	ye
BODY	0	BLOCKAGE	grandprix	on	yes	ye
R-FRONT	1	BLOCKAGE	frnt_whl	on	yes	ye
L-FRONT	2	BLOCKAGE	frnt_whl	on	yes	ye
R-BACK	3	BLOCKAGE	back_whl	off	yes	ye
L-BACK	4	BLOCKAGE	back_whl	on	yes	ye
CO2-CAN	5	BLOCKAGE	co2_can	on	no	ye
INLET	6	INLET	cube3t	off	yes	ye

Select the body and wheel objects. (Left click on each while holding the ctrl key down). Right click on one of the selected items and left click 'surface contour'. Click on **domain** to deselect the body and wheels and show the surface contour clearly.




## 6.7 Streamline plots

This allows us to see how the flow interacts with the body using streamlines. A streamline is the path a dust particle would take if released at that point.

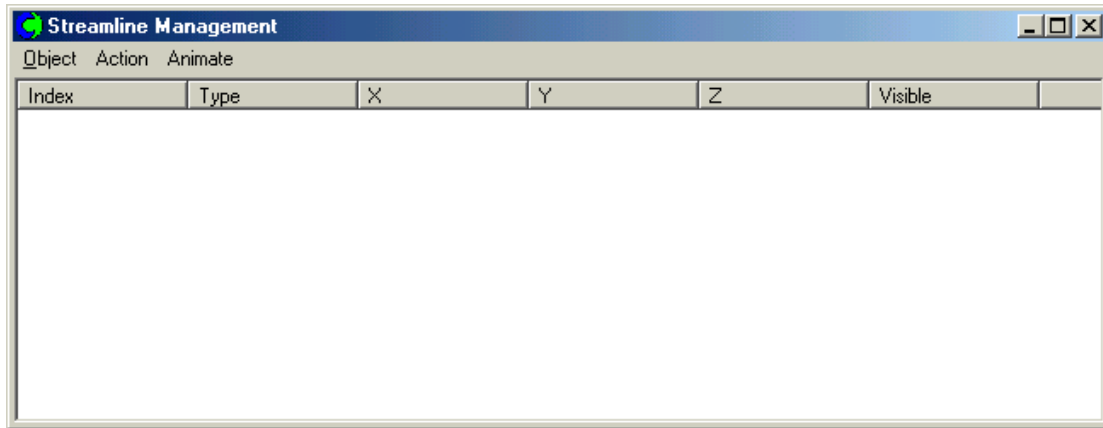




This is achieved by moving the probe to the start point of the streamline, and clicking the streamline button .

- Move the probe to (0.8, 0.25, 0.02) and select the X plane. Click the streamline button.

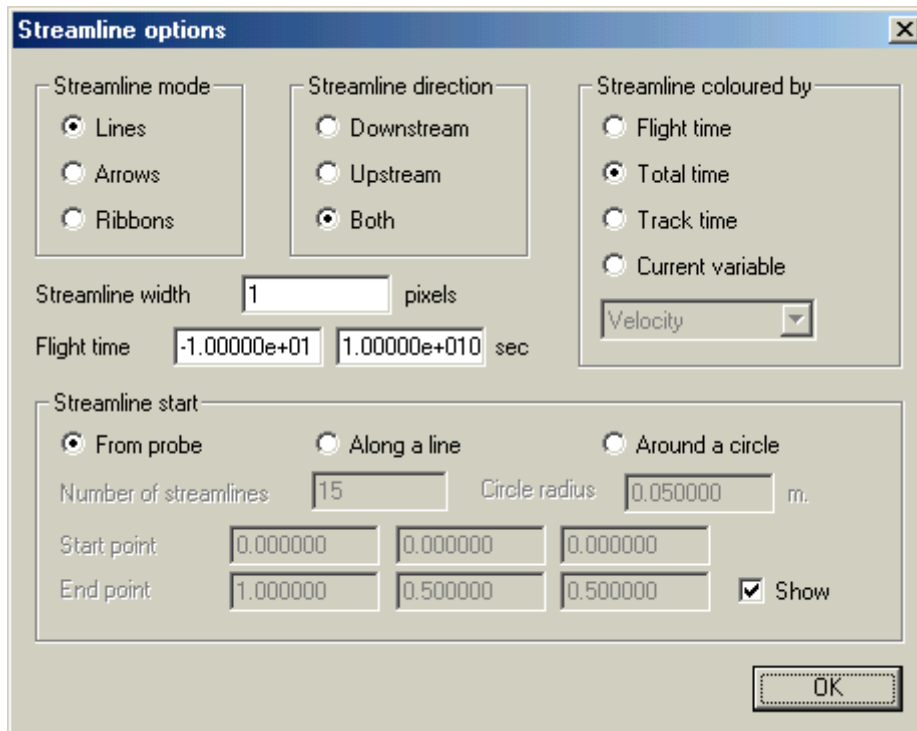
A box saying Streamline Management comes up





- Click on **Object** then select **New** and a streamline is generated, starting from the current probe position. To move the start-location of an existing streamline select the streamline in the Management panel, click on **Action** then **Streamline location**. The streamline start point can be changed. The streamline will adjust to the new start position.


A group of streamlines can be started together.

- Click on object then options or right click , and a dialog like this appears;



Select the '**Around circle**' option, enter a radius of **0.015 m** in the '**Circle radius**' box. You should see 15 balls in a circle, showing where the streamlines will start. Note that the circle lies in the plotting plane, selected by . Click '**OK**'. Click the streamline button , then **Object** and **New**. 15 new streamlines will be generated.

More positions can be selected to give more streamlines in different areas.


To delete streamlines, click the streamline button , click **Object** then **Select all**. Click **Action** and choose **Delete streamline(s)**.


You can experiment with using

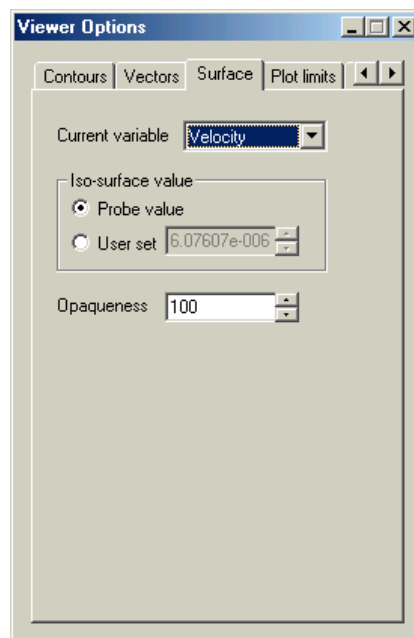
- Lines, arrows and ribbons
- Time (display the time variable)
- From probe, along a line or around a circle

### 6.8 Iso-surfaces

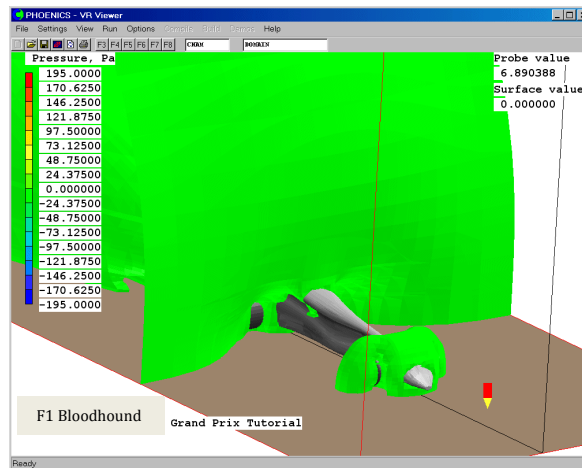
An iso-surface is a surface in space through points of the same value of temperature or pressure or velocity. This can be used to find regions of high or low pressure.

If you left click on  an iso-surface appears. It is for the value of the current variable at this probe position. It will change as the probe is moved.

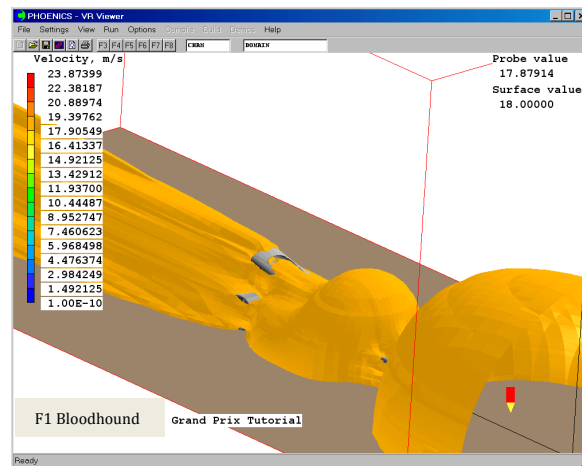
If you right click on  a window like the one below appears



You can then change the variable to pressure or any other variable. You can also specify a particular value for the iso-surface. This iso-surface can be made transparent by setting the opacity to say 50%.



The picture above shows the iso-surface with the variable being pressure, with a value of 0.0 Pa. There is a low-pressure bubble around the front wheels, and another low-pressure zone around the back of the car. High pressure ahead of the car and low pressure behind the car both act to slow it down.



This shows the iso-surface with velocity being the variable, with a value of 18m/s – the running speed. Iso-surfaces of velocity can be used to find high or low speed regions. This can help the designer in the next design phase.

## 6.9 Macros

There are default-view macros already set up in the post-processing VR-Viewer. The choices of macro include:

- Velocity contours
- Pressure contours
- Velocity vectors
- Streamlines
- Iso-surface

Simply click on the down-arrow on  to access these.

For further information about creating your own macros, click on:

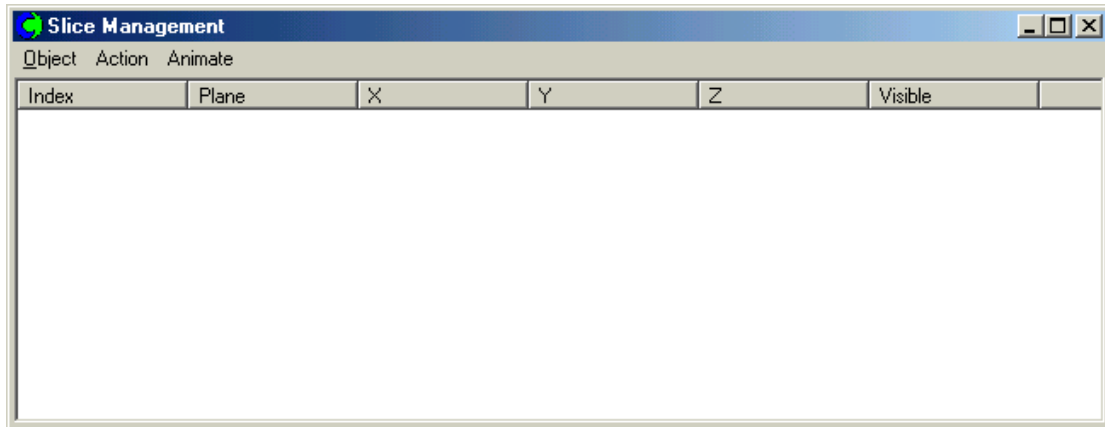
[http://www.cham.co.uk/phoenics/d\\_polis/d\\_docs/tr326/vr-view.htm#VR-Viewer Scripting Facility](http://www.cham.co.uk/phoenics/d_polis/d_docs/tr326/vr-view.htm#VR-Viewer Scripting Facility).

### 6.10 Slices

Slices are stored contours and vectors for a particular probe position. By storing several slices, you can build up a more complete picture of the flow.

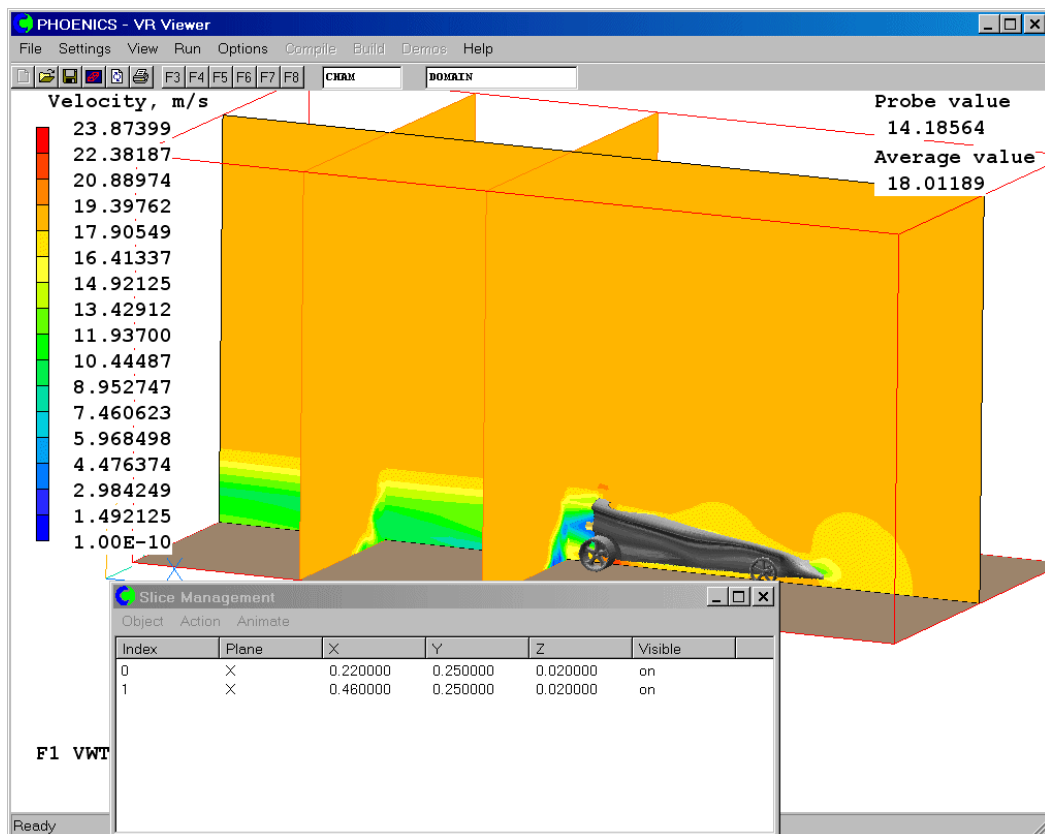
- To save the current probe position and plotting plane direction, click the slice

management button . The window below comes up




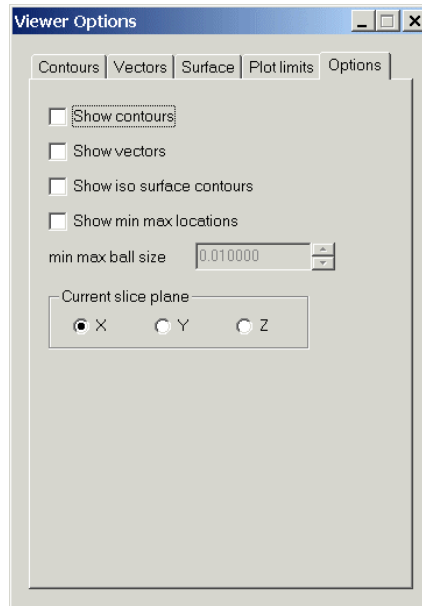
Click **Object**, then select **New**. A slice will be saved at the current probe position. If you now move the probe, you will see two plots, one at the probe and one at the slice. To modify the existing slice, select it from the list in the slice management dialog, then use one of the options on the **Action** menu to change it.

This shows two stored X slices and the Y plotting plane with velocity as the variable.

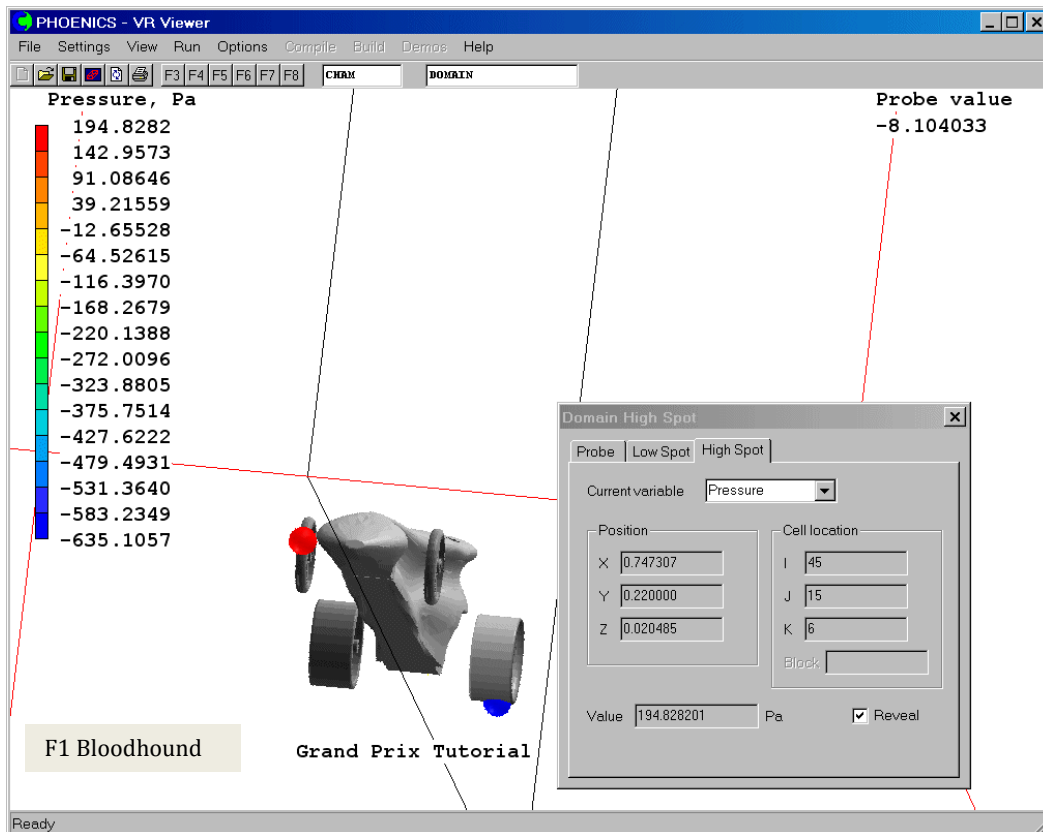


### 6.11 Minimum and Maximum Values

To display the positions of the minimum and maximum values of the current variable, click on , then open the **Options** tab.



Check the **Show min max locations** box. The minimum and maximum locations will be shown by a red and blue ball. To see the actual values, double-click on either the red or blue ball.



The highest pressure is ahead of the front wheel, and the lowest is immediately behind the back wheel.

## 7. Coefficient of Drag and Aerodynamic Forces

The VWT outputs drag force data at the end of every run. This is stored in the \*.res (results file) or in this case the Grandprix.res file.

- Click on '**File**' and select '**Open File for Editing**' in the main Menu Bar.
- Go to '**Result (output file)**'. This will open up a text file.

Scroll to the bottom of this document as shown below.

```

result - Notepad
File Edit Format View Help
*****
Integrated forces for body and wheels
Drag - Fx = -5.577832E-01 N
Side - Fy = 1.097868E-02 N
Lift - Fz = -1.372610E-01 N

Force unit vector: -9.708536E-01 1.910902E-02 -2.389105E-01

The total force acts at:
X = 5.848188E-01 m
Y = 2.498072E-01 m
Z = 2.391274E-02 m

The loadings at the front and back axles are:
Front = -4.166697E-02 N ( 4.247397E+00 gm)
Back = -9.559400E-02 N ( 9.744545E+00 gm)

Normalisation area = 5.291234E-03
Free-stream dynamic head = 1.926180E+02

Integrated Drag coefficients

Drag - Cdx = 5.472826E-01
Side - Cdy = 1.077200E-02
Lift - Cdz = 1.346769E-01

*****
*****
SATLIT RUN NUMBER = 1 ; LIBRARY REF.= 0
Run completed at 16:29:20 on Wednesday, 10 January 2007
MACHINE-CLOCK TIME OF RUN = 1900 SECONDS.
TIME/(VARIABLES*CELLS*STEPS*SWEEPS*ITS) = 1.396E-05
*****

```

The values for force and drag coefficient are split up into different components of the car (Body, wheels and CO<sub>2</sub> canister). The image shows the printout of the total (integrated) forces summed over all components.

The bigger the CD values the larger the force will be on that component.

The X-direction forces are in the direction of motion, and represent drag. Negative values of Fx represent the force pushing the car backwards along the X axis.

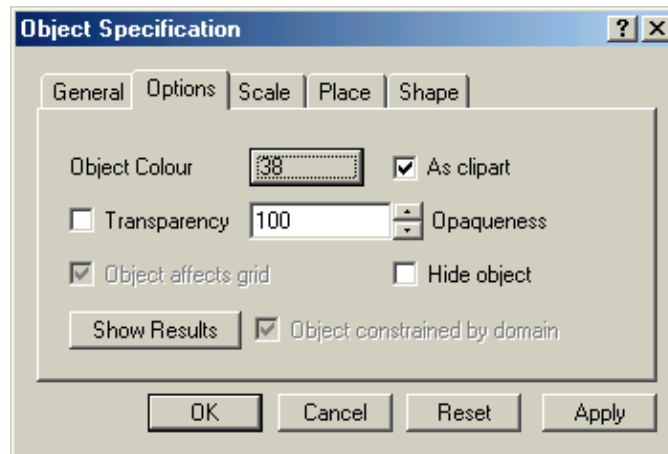
The Y-direction forces are in the side-to-side direction, and the integrated value should be close to zero due to symmetry. In this example the total Fy is 50 times smaller than the drag force, Fx.

The Z-direction forces are in the vertical direction, and represent lift. A negative total Fz acts downwards pushing the car into the road. A positive force would try to lift the car off the ground – if this force were bigger than the weight of the car, it would take off!

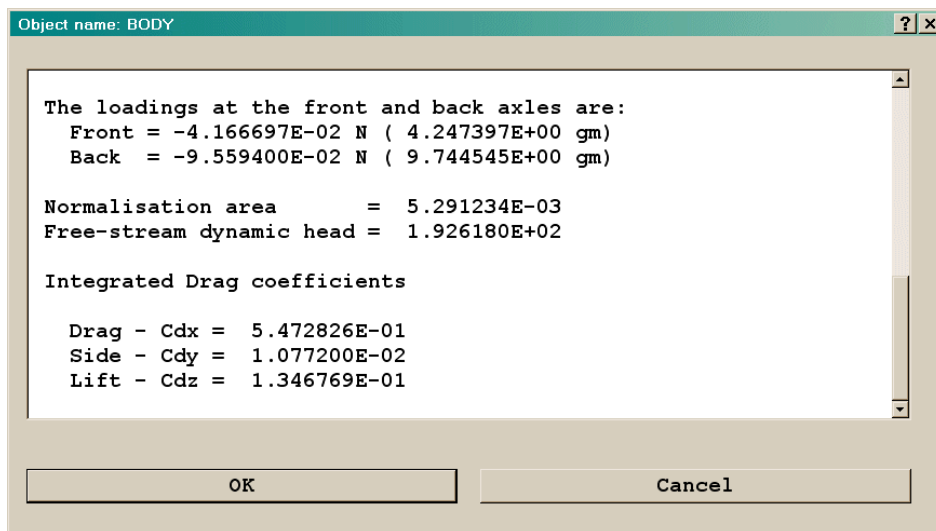
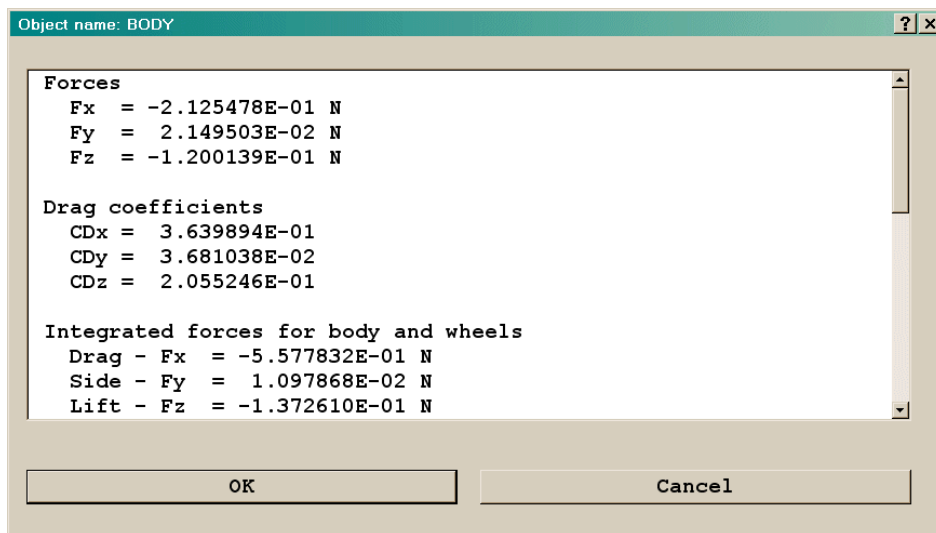
You will notice that the wheels contribute much of the drag force.

You will also see the loading at the front and rear axles. These are the readings you would see if the car was standing on two scales, which had been zeroed with no flow. Positive numbers indicates a down force, negative numbers indicates lift.

These values can also be seen in the VR viewer by double clicking on the BODY object, select the **Options** tab and clicking **Show Results**.



The following display will appear:



This concludes this introductory tutorial into the VWT. For further information about the VWT and its principles please read "The Balsa Wedge" demonstration case.

## Virtual Wind Tunnel – Tutorial – Formula One Class

The steps to be followed are exactly the same as for the Bloodhound:

1. Generate the car body shape, then export an STL file. In this example, the STL file is called 'R6 F1 Car Template.stl'. The F1 STL import software will assume that the largest dimension is along the length of the car body, and that the body width is greater than the height. The STL file's units are assumed to be millimetres.

If you have trouble with generating this body shape, a copy of the R6 F1 Car Template.stl file can be found in `\phoenics\d_intfac\d_cadpho\d_stl`. Copy this file to `\phoenics\d_priv1`, and proceed with the tutorial.

2. Start F1 VWT, click on File, Start New case, and select F1-FormulaOne. The Formula One balsa blank will appear.

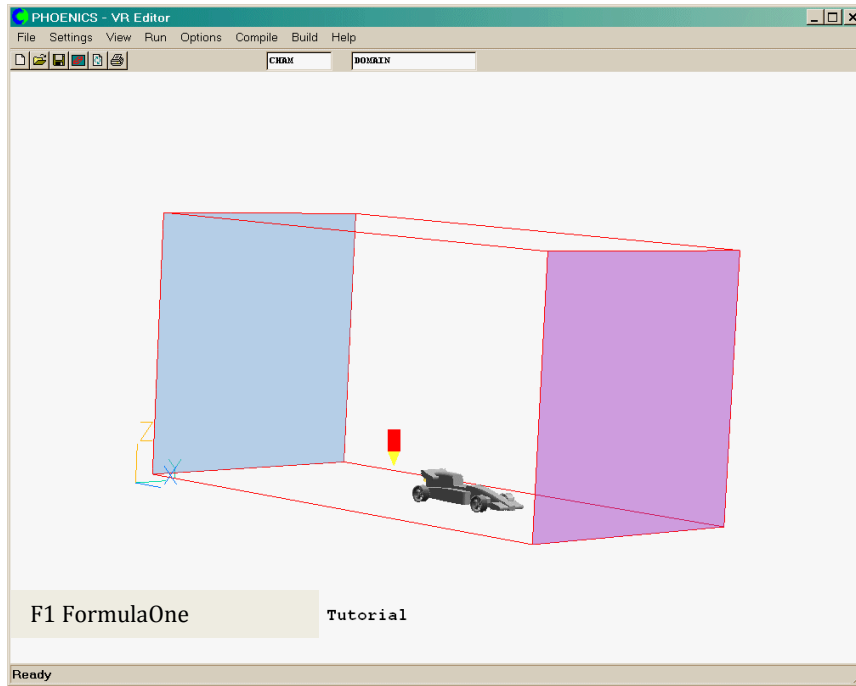


3. Click on '**Menu**' located on the VR headset, and change the title of the simulation to 'Formula One Tutorial'
4. Click on  and select the '**geometry file**' option (currently selected as '**balsa-r**'). This will open up a dialog in which the STL geometry file should be selected.
5. Click on ; here the '**Running Speed**' is set, but for the purposes of this tutorial it should be left at **18 m/s** (default). This is equivalent to 59 ft/s in the FPS unit system, which can be chosen at any time.
6. Click on . This window gives you the option to select the grid (coarse, medium, fine or user) and the number of iterations. For this tutorial the following selections should be made:
  - Number of Iterations = 300 (default = 100)
  - Grid = Coarse.

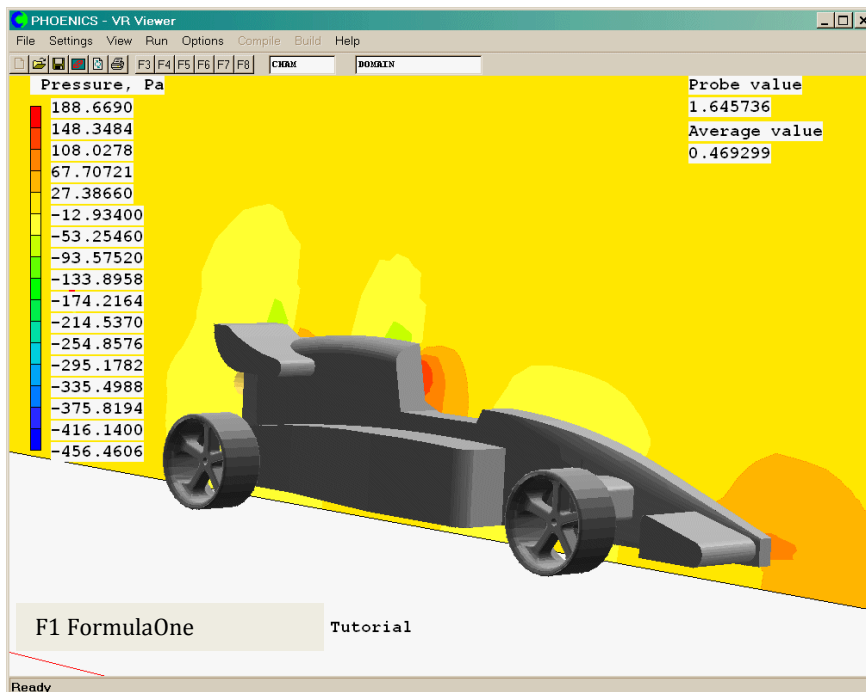
**NOTE:** It is important that the '**Restart from Previous Run**' option is set to '**off**' (default). This window also gives the user an option to switch to full PHOENICS, but this should only be attempted in the presence of a teacher.

7. Click on  and then '**OK**'. This will take you back to the main VR environment. The car shape should change to:



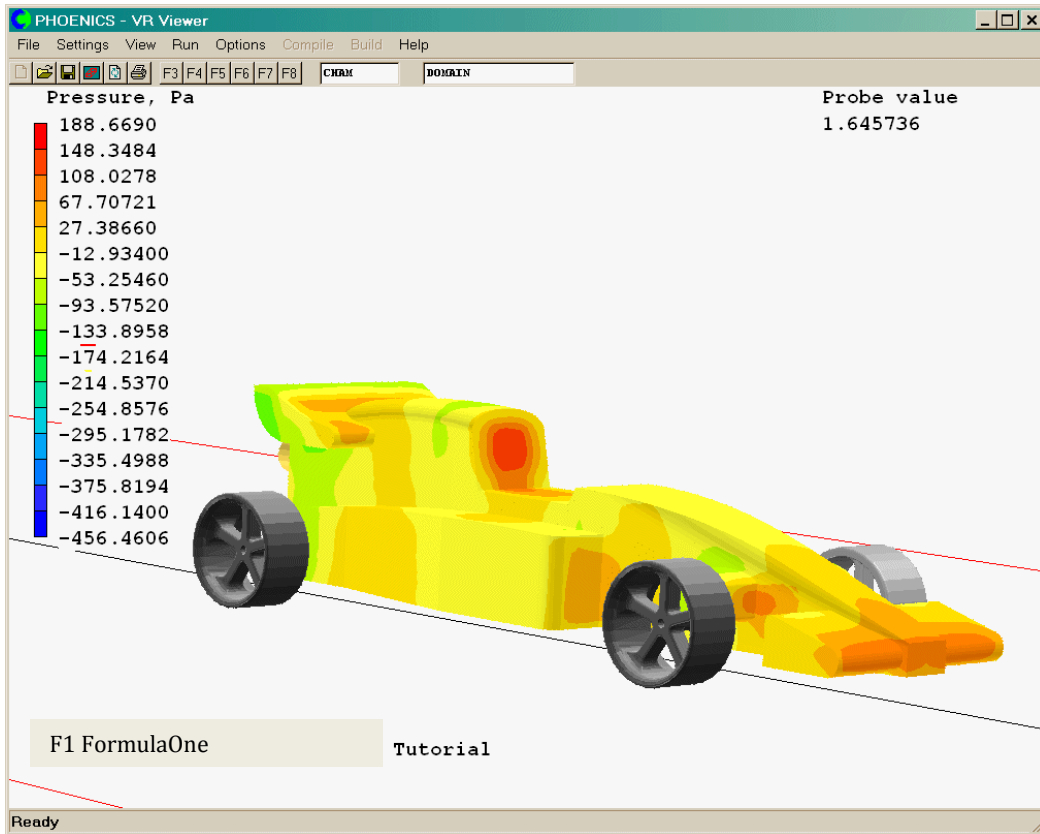


8. Click on 'Run – Solver' to run the solver to obtain the CFD solution. Once this has finished,
9. Click on 'Run – Postprocessor' to view the results. For detailed suggestions on how to do this, refer to the F1 Bloodound tutorial. A few sample images are shown here as examples of what can be seen.

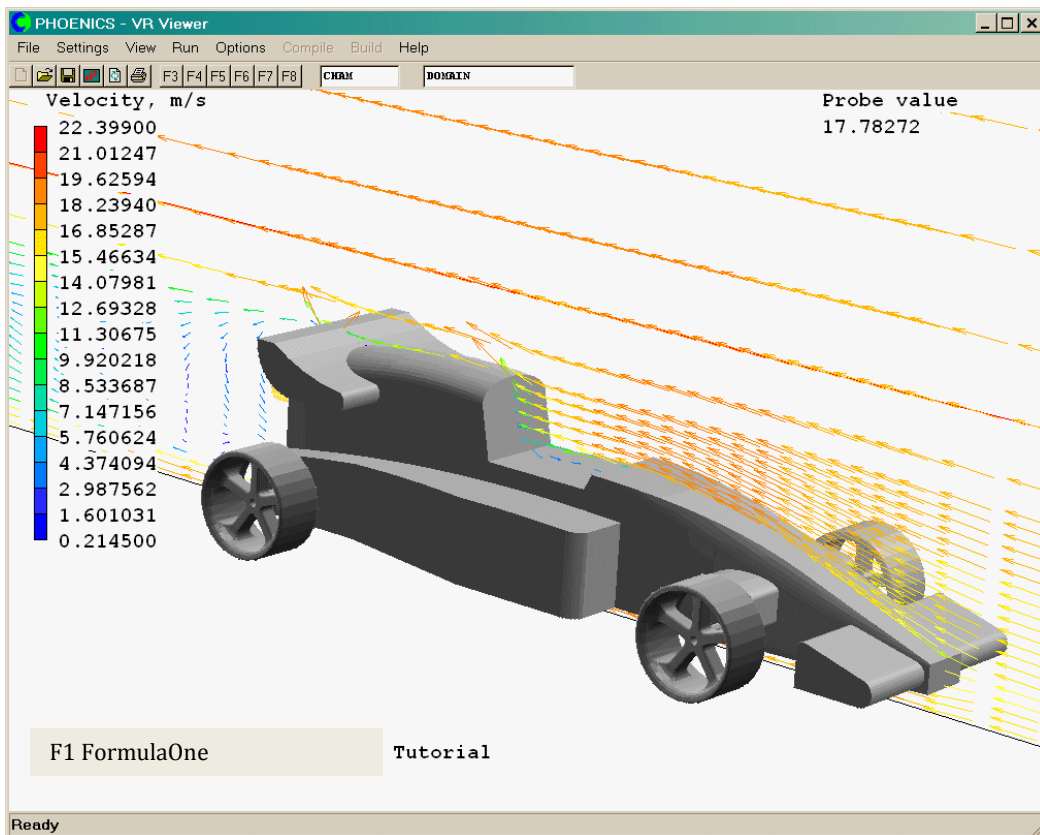


Pressure on Y-plane (Y=0.25)

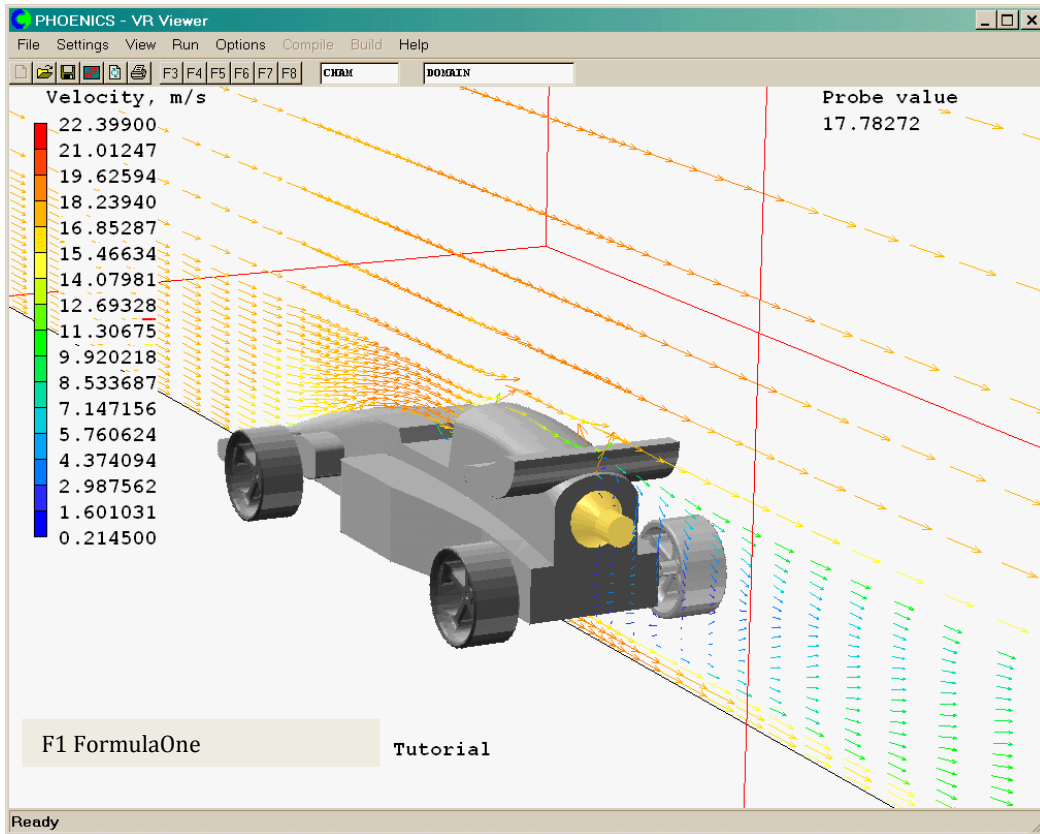
The high pressure (red area) on the back of the cockpit shows a region producing high drag.



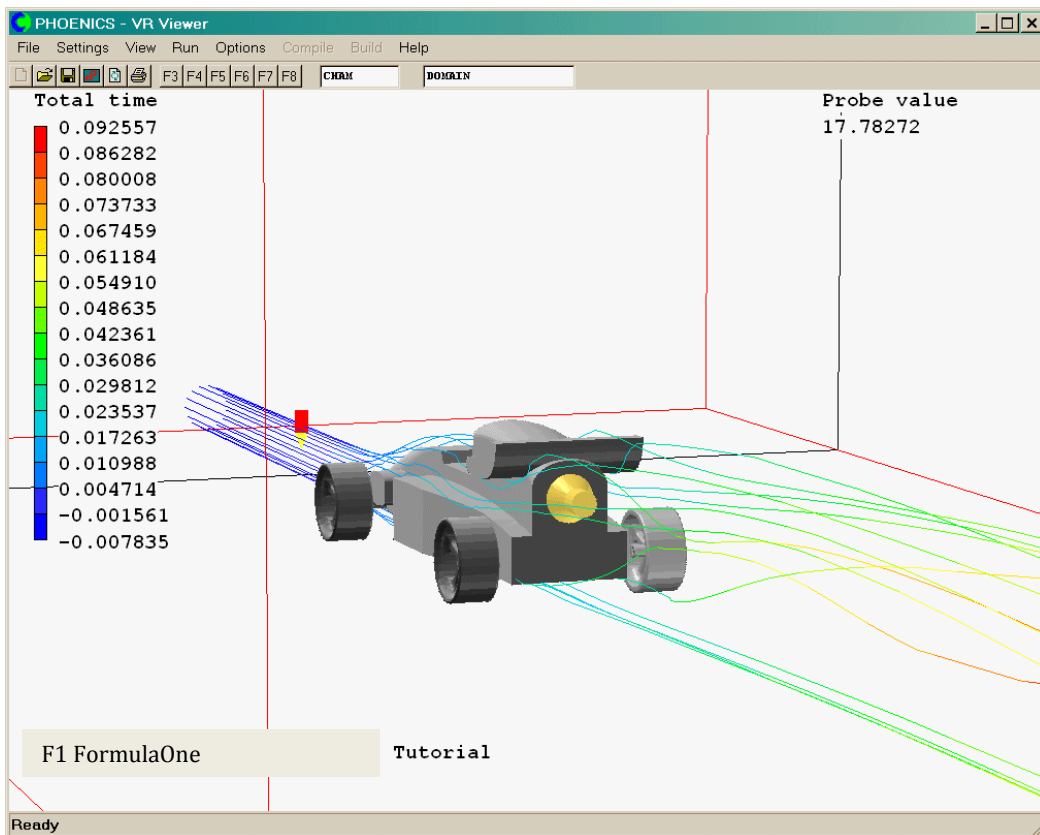
Pressure surface contour



Velocity vectors on Y-plane (Y=0.25)



Velocity vectors on Y-plane (Y=0.25)



Streamlines started around a circle on X plane

## Demonstration Case - The Balsa Wedge

### 1. Introduction

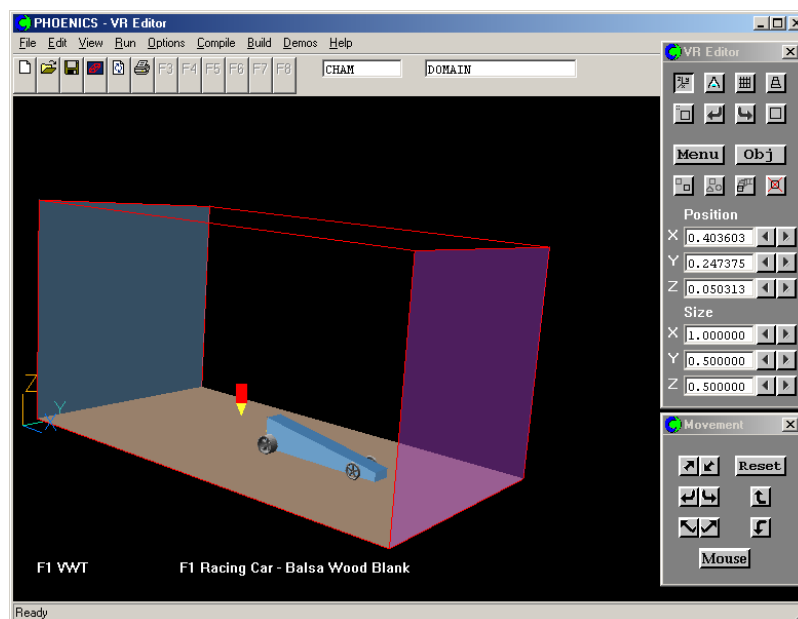
This demonstration case is designed to highlight further the versatility of the VWT. Although not an instructional example like the Grand Prix tutorial the balsa wood wedge simulation will show the following:

- The importance of grid refinement.
- Aerodynamic improvements gained through modeling.

The structure of this demo will be to compare a 'coarse' and 'fine' meshed wedge example, followed by a comparison with the 'fine' Grand Prix case. This case was run with F1VWT MKII, so some of the screen images may no longer reflect the exact appearance of the current version.

### 2. Geometry, Setup and Running

The coarse and fine grids are used with the balsa blank geometry (below). The settings in both cases are left as the VWT defaults, the only variables altered are evidently the grid and number of iterations.



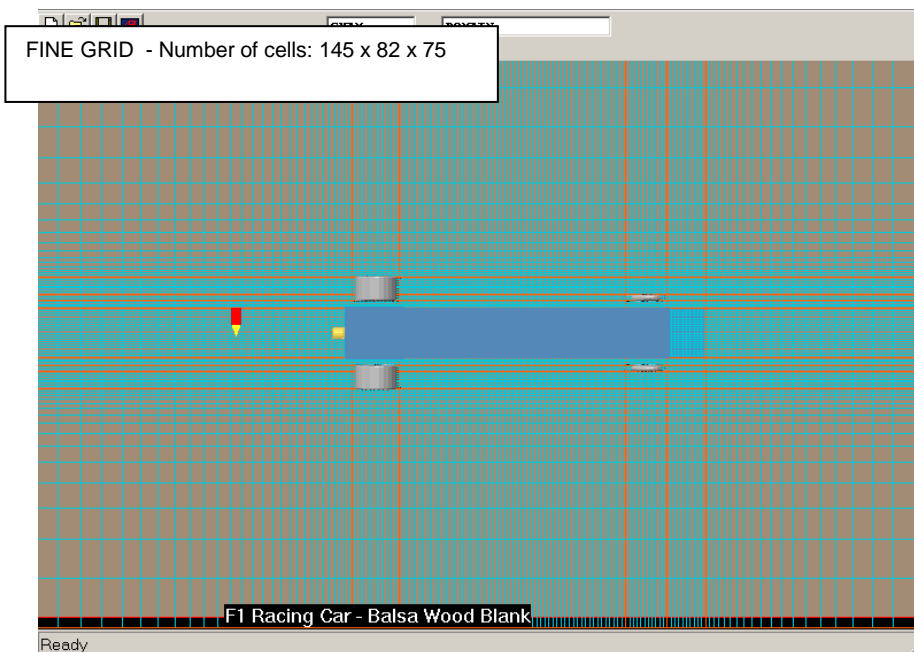
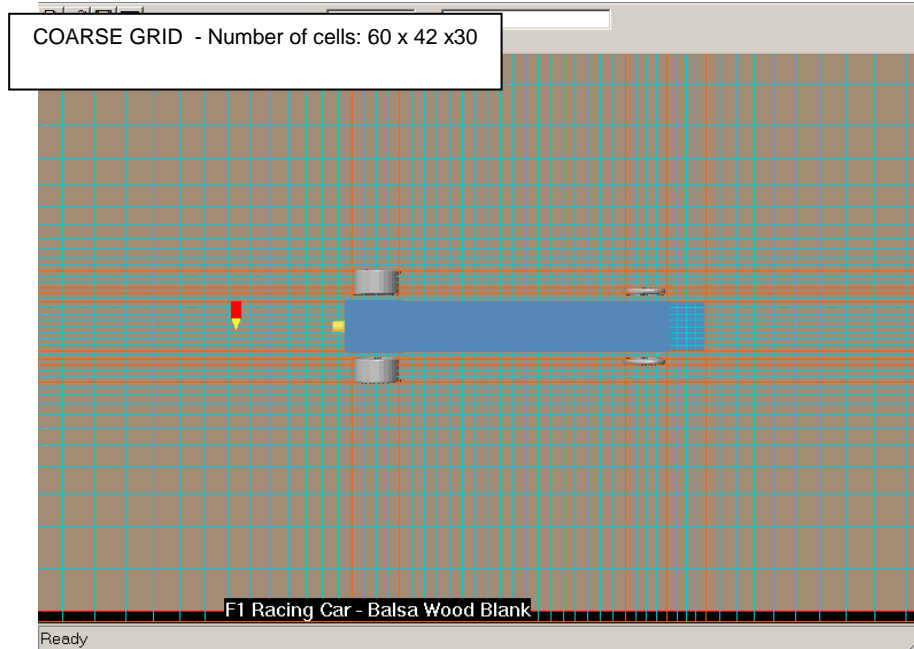
For the coarse grid, **600 iterations** will prove sufficient and **1000** shall be required for convergence of the fine grid. The probe, as in the tutorial, is moved in to an appropriate position behind the car body.

The solutions are run to convergence giving similar graphs to that shown in the tutorial. The run times using a 1 GHz PIII computer with 500MB of RAM are:

- 40 minutes for the coarse grid.
- 14 hours using the fine mesh.

**Note:** The medium mesh although not shown in this demonstration case will require approximately seven hours to converge using this machine.

It is important to understand the difference between the coarse and fine grids. The following two plots show first the coarse and then the fine mesh, in plan view.

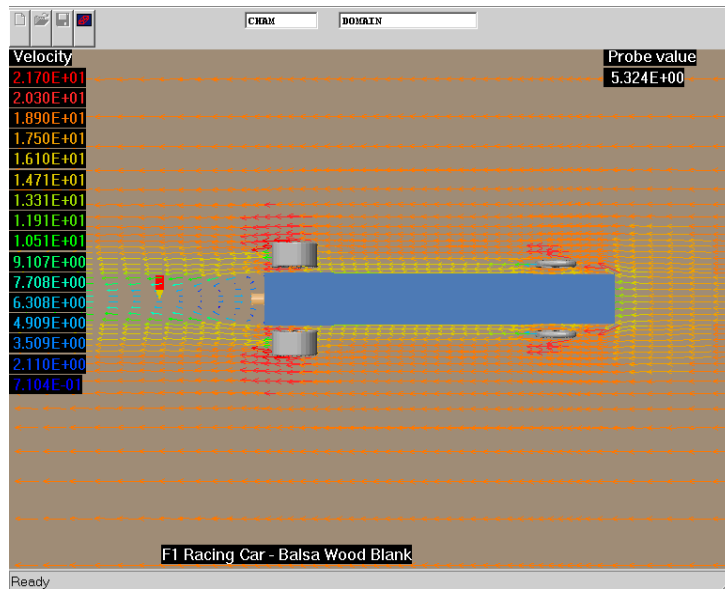


The boxes show how many cells (x, y, z) make up the grid in each case. The fine grid has significantly more cells than the coarse grid. Notice the concentration of cells directly around the car in both cases.

Now that the grid differences are understood the next step is to see how they translate into results.

### 3. The Results

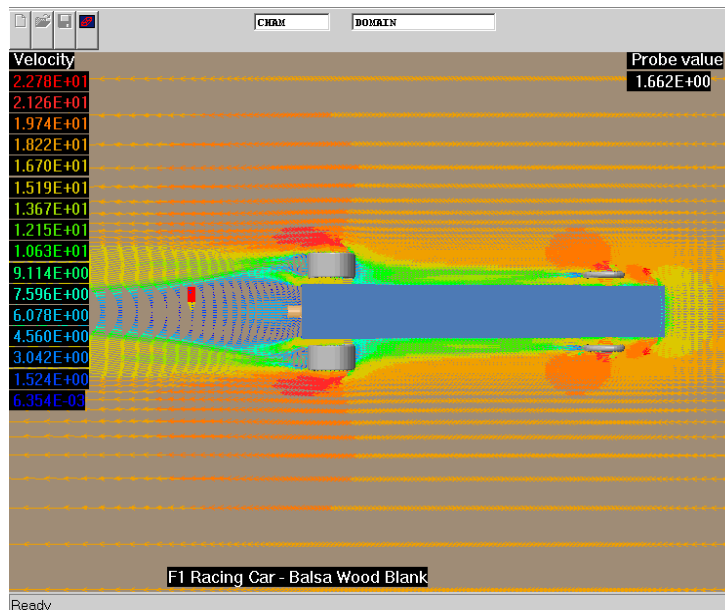
This section will directly compare the fine and coarse results and thus showing the greater definition gained when using a finer mesh.



The coarse grid shows approximately what is happening around the car, the finer mesh has much more clarity.

For example, the wake behind the 'coarse mesh' car is small and undefined. Equally, behind the wheels there is a distinct lack of definition.

Although, the coarse mesh captures occurrences such as the velocity increasing around the wheels, overall the large cell-spacing inhibits the flow solution.

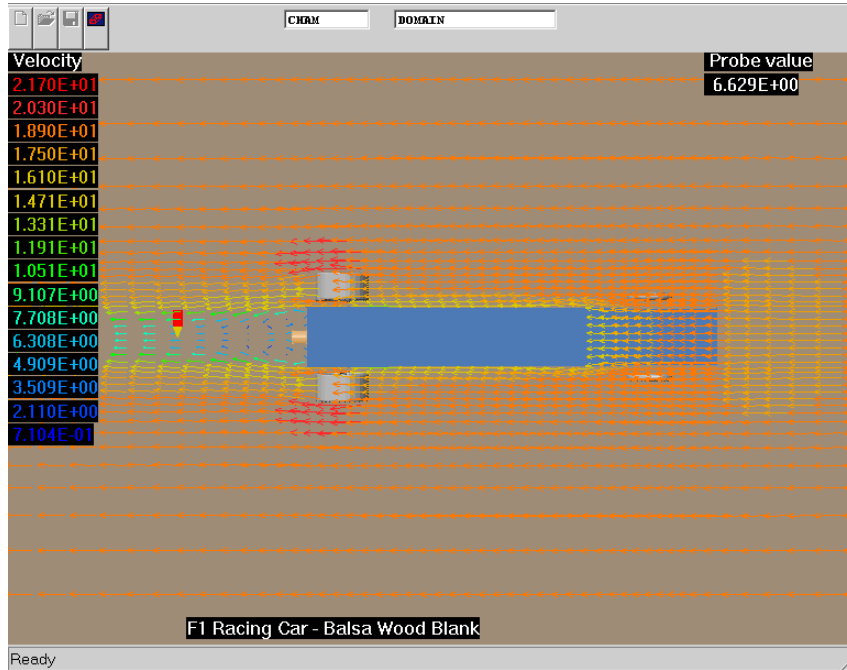


The convenience of having a fast run time cannot be ignored if a general impression of the flow is all that is required, but the large increase in definition shown in the second plot (fine grid), gives a much clearer idea of the real flow regime.

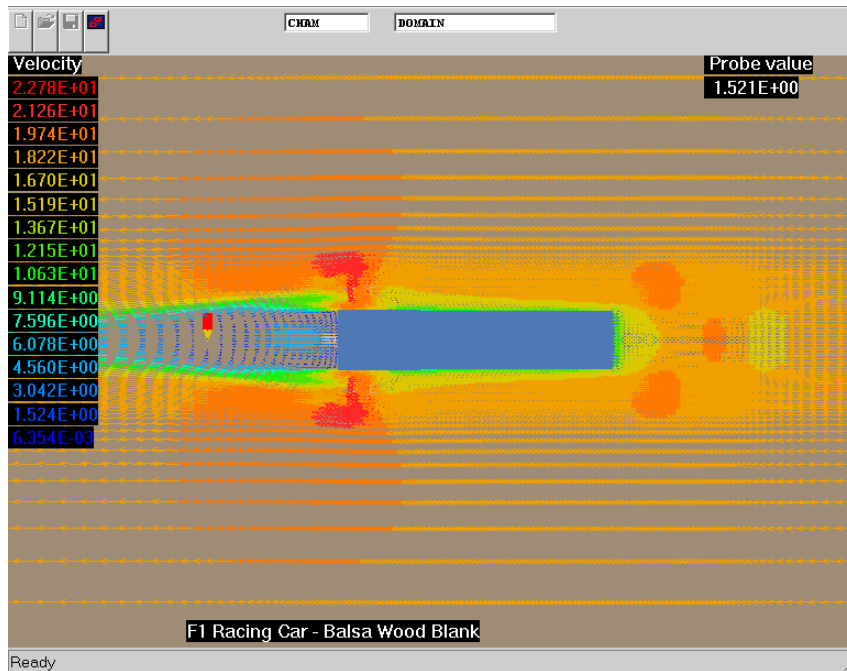
These plots are both taken in the Z-plane at Z=0.02.

The two plots shown below are again taken in the Z-plane but at Z=0.04. The coarse plot shows very little effect of the wheels, in comparison to the fine mesh where the flow is heavily affected. This is due to there being very few cells around the wheels using the coarse grid, there are of course many more with the fine mesh.

**COARSE GRID**

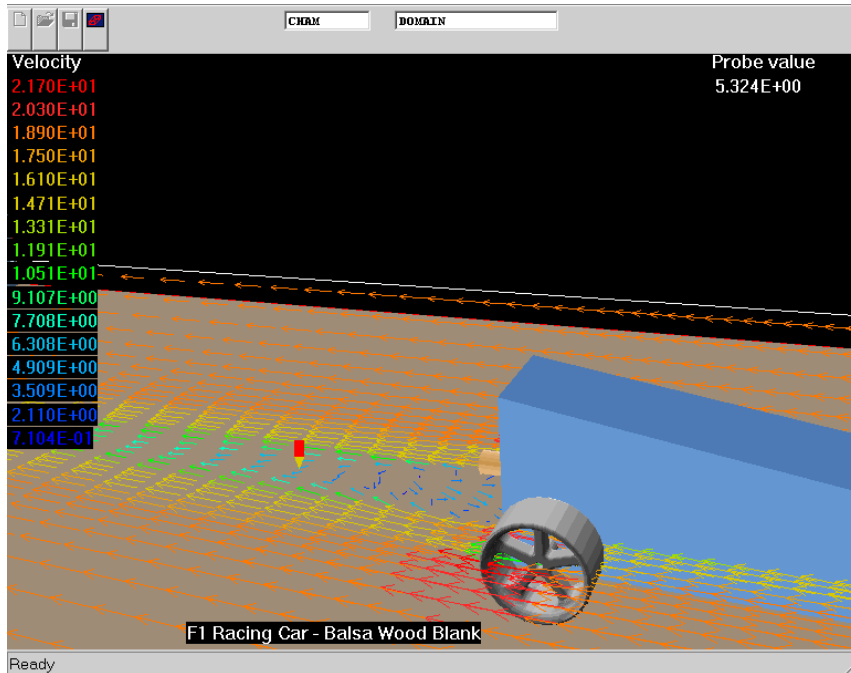


**FINE GRID**

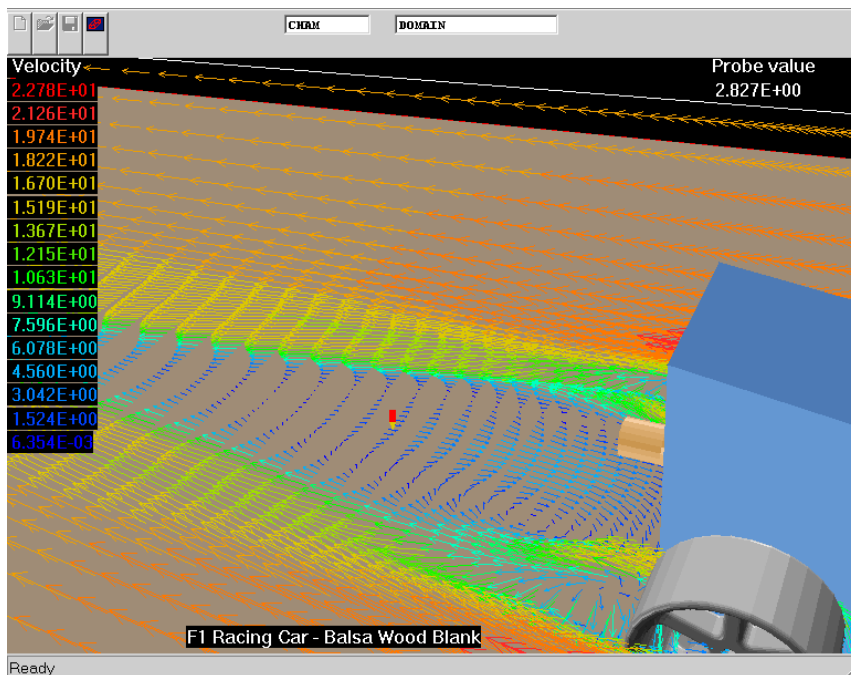


Focusing on the wake behind the car body shows that whilst the overall effect is indicated using the coarse grid, the fine mesh clearly captures areas of recirculation. The wake behind the wheels can be seen with the fine mesh but not with the coarse grid.

**COARSE**



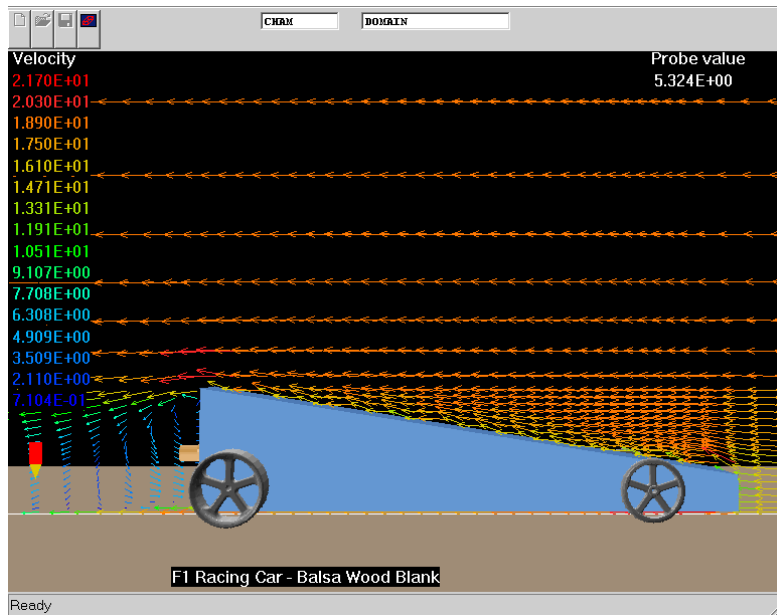
**FINE**



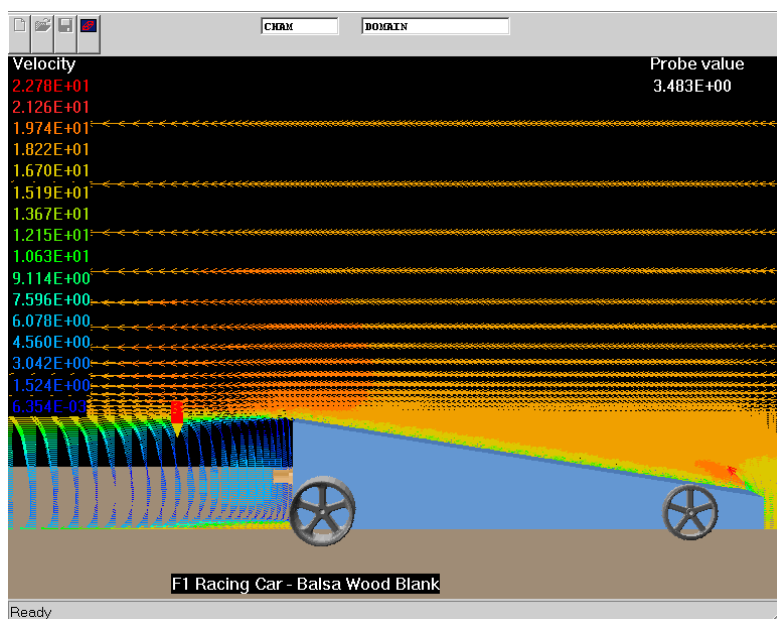


These plots show the profile views of the car. An interesting point to note in the coarse grid plot is the step-effect along the top of the car, this is due to lack of cells and so the angular surface is not picked up accurately. The fine grid does not have this problem the increased number of cells has picked out the surface more accurately. (Note: For industry, "Body-Fitted, "Unstructured"" or "Adaptive Grid" meshes can also be used but these do not form part of the standard F1 VWT product.)

**Coarse**



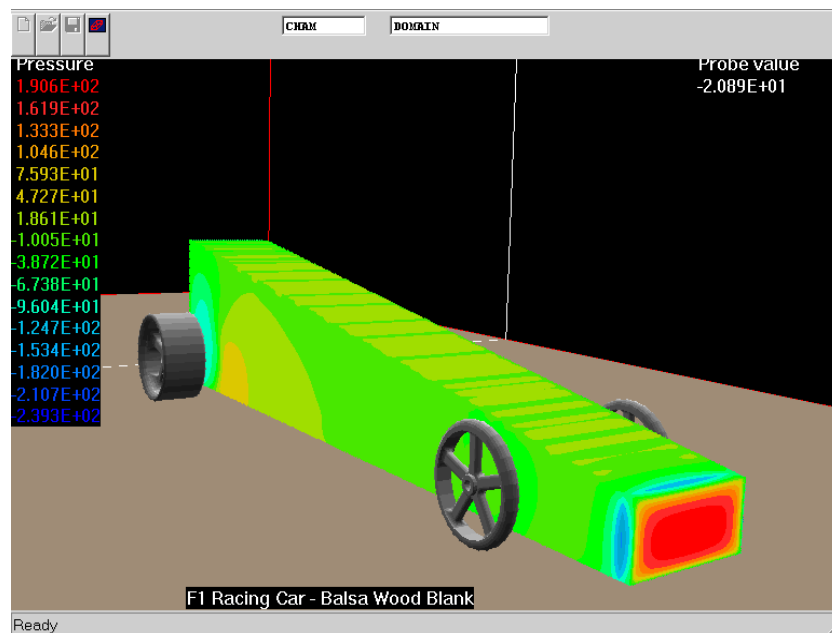
**Fine**



#### 4. Increase in aerodynamic efficiency

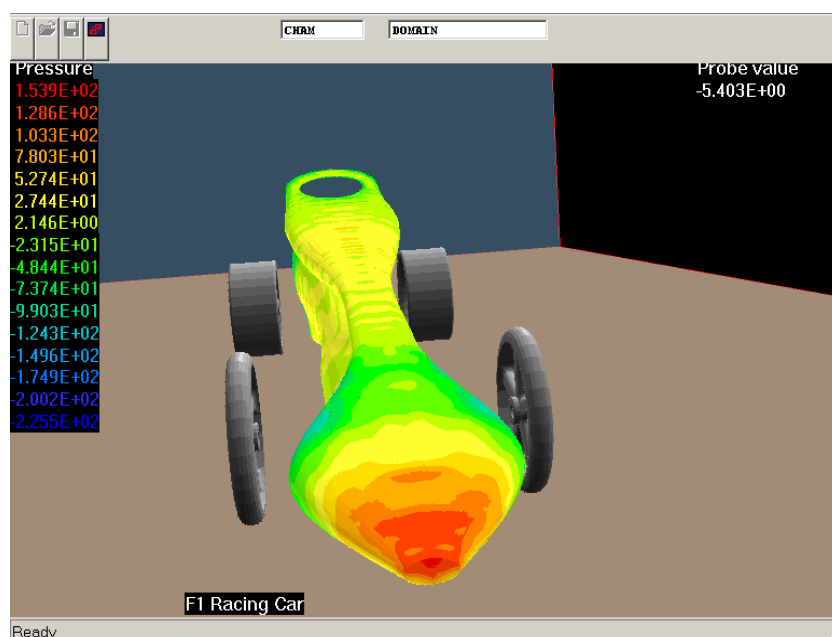
The grid comparison shown above has hopefully highlighted important 'pros and cons' of using a coarse or fine grid. From this point, the demonstration will concentrate on the second objective of proving the increase in aerodynamic efficiency. This will be done by comparing the fine 'wedge' simulation to a fine 'grand prix' simulation.

The most important aspect to look at is the car's body shape. This causes most of the drag and, being the most changeable part of the car, is crucial to any design.



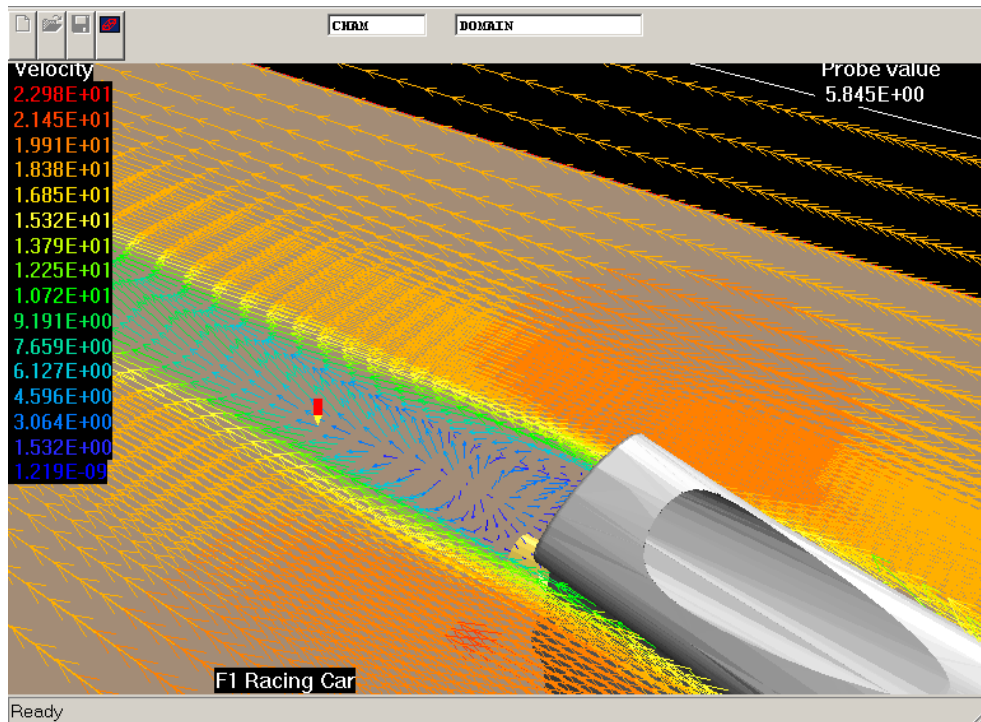
The first picture shows the wedge shape with pressure contours plotted on to its surface. The most noticeable aspect is the large pressure on the nose of the car.

This is where the flow is reduced to zero velocity; meaning that the energy within the flow is working directly against this surface area moving through space. As a result the car is less efficient.

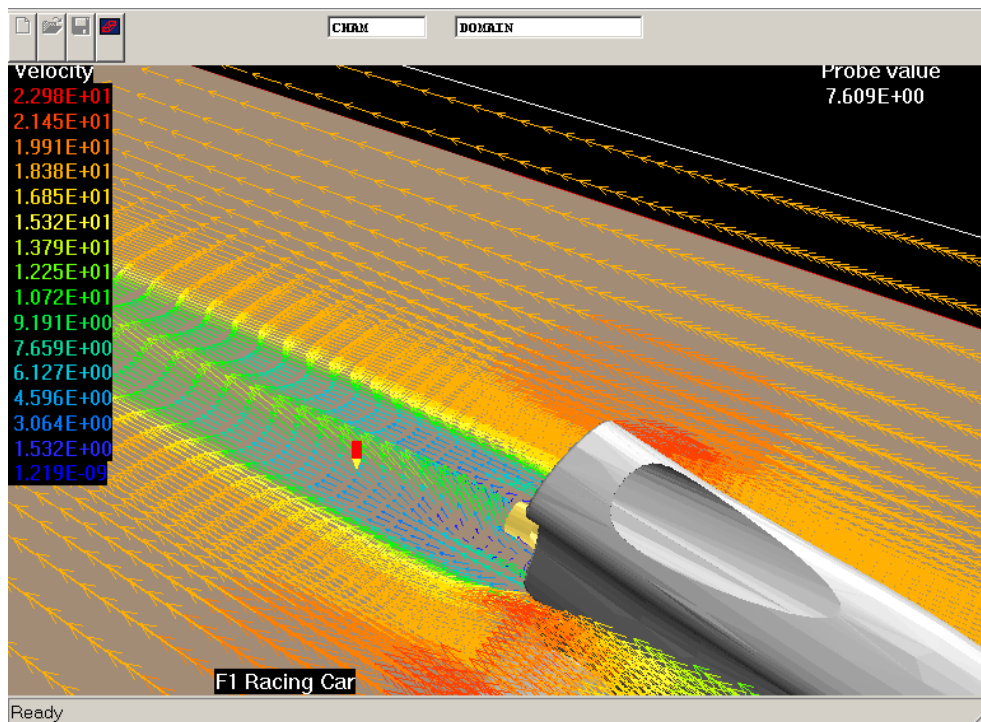


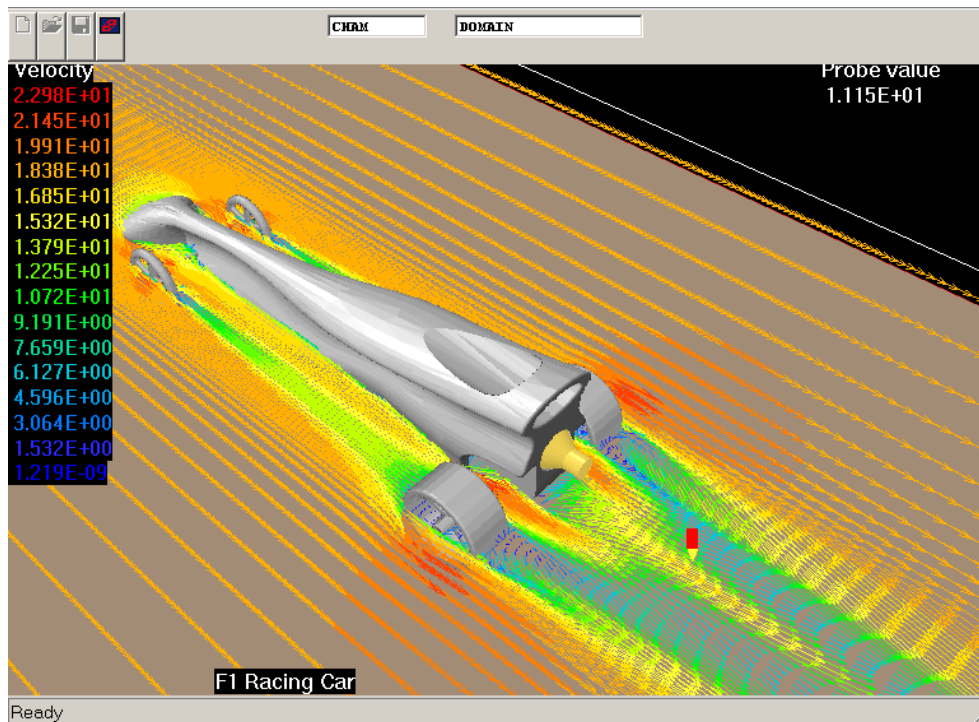
The second plot shows a Grand Prix car example. The pressure build-up is not as significant as with the bluffer wedge shape, due to the shape of the nose. This is primarily due to the reduction in surface area compared to the wedge, the projected surface is still quite large in this case and although the design is better, it is still not an ideal solution.

The wake left behind the car is also significant in drag generation. The extent of recirculation behind the wedge shape body has already been shown. The next three plots below show the Grand Prix car wake.

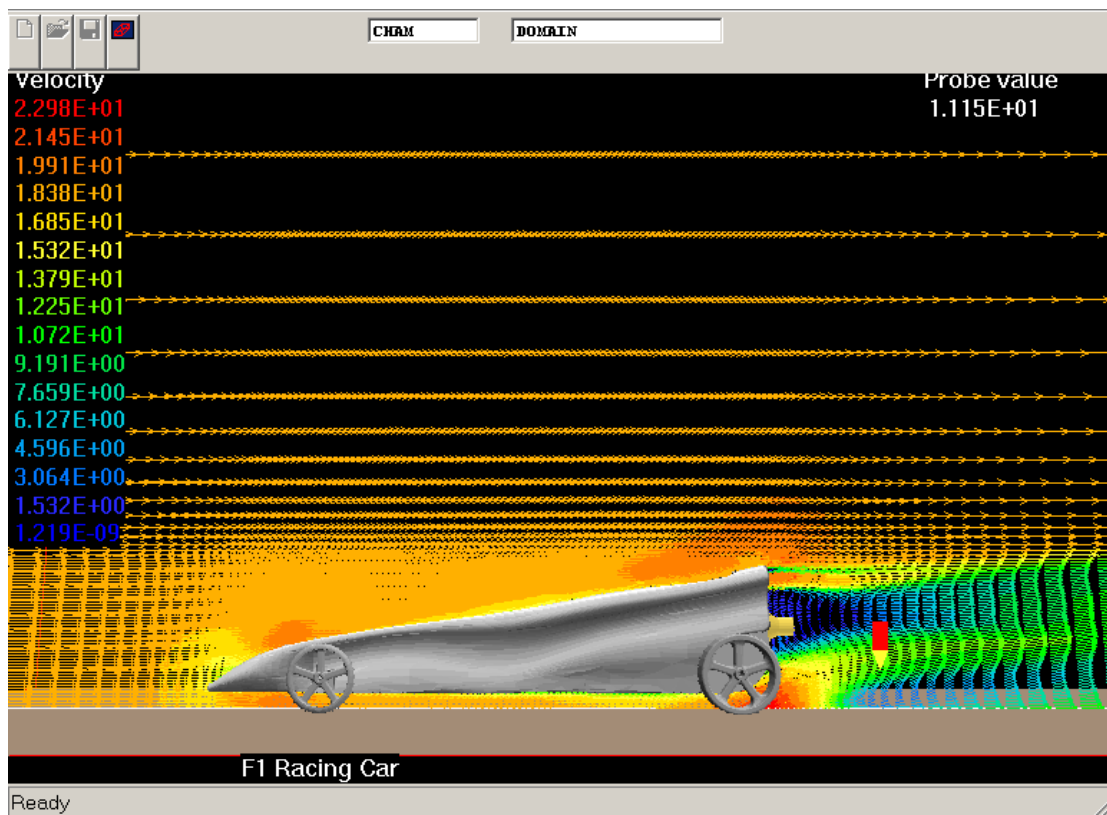


When compared to the wedge shape shown previously, the wake behind the body is significantly less.





This third plot shows clearly that the distinct re-circulation regions, present with the wedge, do not occur with this design. Note that the rear wheels generate a large wake. This is the same in both examples.



This last plot again emphasizes the effect the Grand Prix body design has on the wake. The fluid can also be seen to accelerate through the hole at the rear of the car.

## 5. The drag forces

Another aspect of the **VWT** (as explained in the tutorial) is extracting the drag force data stored in the **\*.res** file. The table below shows the comparison between the wedge and Grand Prix bodies.

Balsa Wedge (body only)		Grand Prix (body only)	
$F_x$	-2.709499E-01 N	$F_x$	-2.465826E-01 N
$F_y$	1.458093E-03 N	$F_y$	-8.874801E-03 N
$F_z$	-2.537223E-02 N	$F_z$	-1.040585E-01 N
$CD_x$	4.925315E-01	$CD_x$	3.972756E-01
$CD_y$	5.735270E-04	$CD_y$	3.348176E-03
$CD_z$	1.045421E-02	$CD_z$	3.885528E-02

The values for  $F_x$  (highlighted in red) are lower for the Grand Prix car, this is as expected due to its more aerodynamic shape. In conjunction with this the coefficient of drag in x ( $CD_x$ ) (highlighted in blue) is significantly higher for the wedge shape.

The final point to mention is the CD and forces in the Z direction. They will effectively represent down force generated by the car. These values are larger for the Grand Prix car.

Within the results file is the same data for each object in the domain. The following table shows the drag components generated by the back wheels.

Wedge (R-wheel only)		Grand Prix (R-wheel only)	
$F_x$	-1.442101E-01 N	$F_x$	-1.234675E-01 N
$F_y$	-6.596514E-03 N	$F_y$	7.090292E-04 N
$F_z$	8.877212E-02 N	$F_z$	8.999211E-02 N
$CD_x$	4.248290E-01	$CD_x$	3.637232E-01
$CD_y$	4.919665E-02	$CD_y$	5.287906E-03
$CD_z$	2.625875E-01	$CD_z$	2.661964E-01

As is expected the difference in CD and F for the same wheel on the two models is slight, with the only difference coming in the y direction but this can be attributed to the different body designs.

From this data it is evident that the wheel adds significantly to the drag of the car, particularly when looking at the values of  $CD_x$  and  $F_x$ .

This concludes this demonstration of the VWT. Further information can be gained by reading the "Jaguar F1 Car" demonstration case.

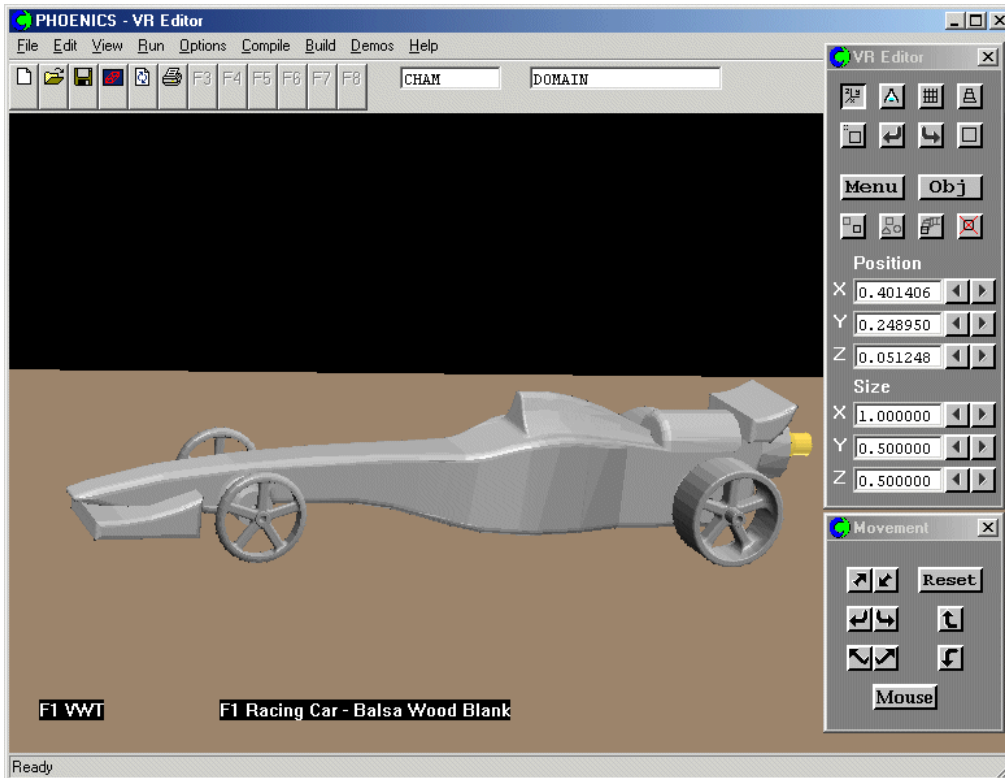
## Demonstration Case – Jaguar F1 Car

### 1. Introduction

The simulation of the Jaguar F1 car was performed using the **F1 VWT MKII**, so some of the screen images may not exactly reflect the appearance of the current version. The following document outlines the setup and results gained from the computation.

### 2. The Geometry

The geometry of the Jaguar car was set up as in Plot 1. The STL file can be found as `\phoenics\d_intfac\d_cadpho\d_stl\jaguar_f.stl`.



### Plot 1

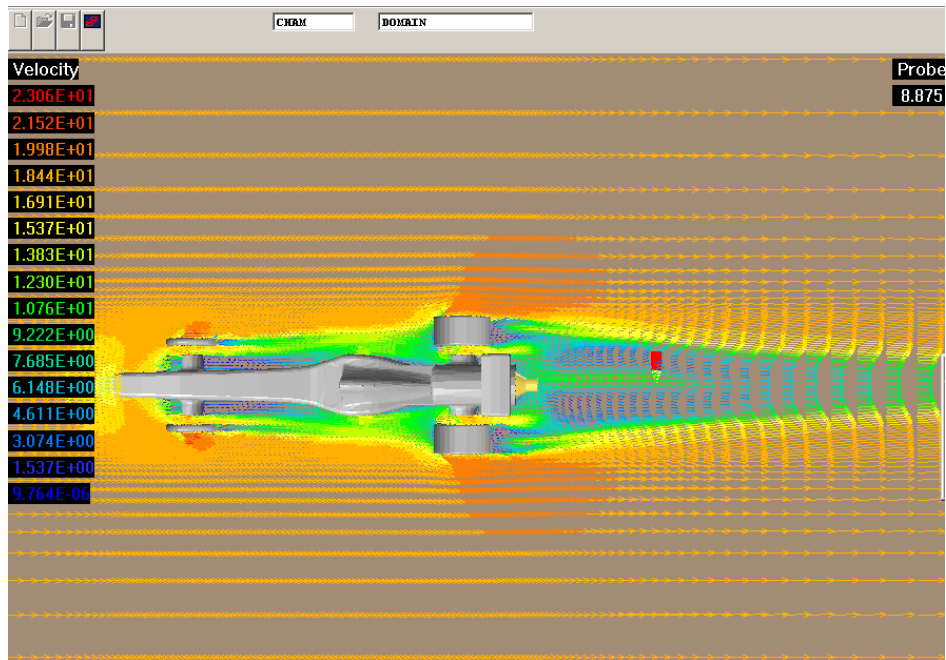
The settings for this case are as follows:

- Ride Height = 0.005.
  - Axle Positions: Front = 0.5, Rear = 0.28.
  - Axle Length: Front and Rear = 0.055.
  - CO<sub>2</sub> Height = 0.024.
  - Wheel Diameter: Front = 0.035, Rear = 0.04.
  - Wheel Width: Front = 0.005, Rear = 0.02.
- The speed of the car is set to 18m/s, with the simulation running for 2000 iterations using the fine grid.

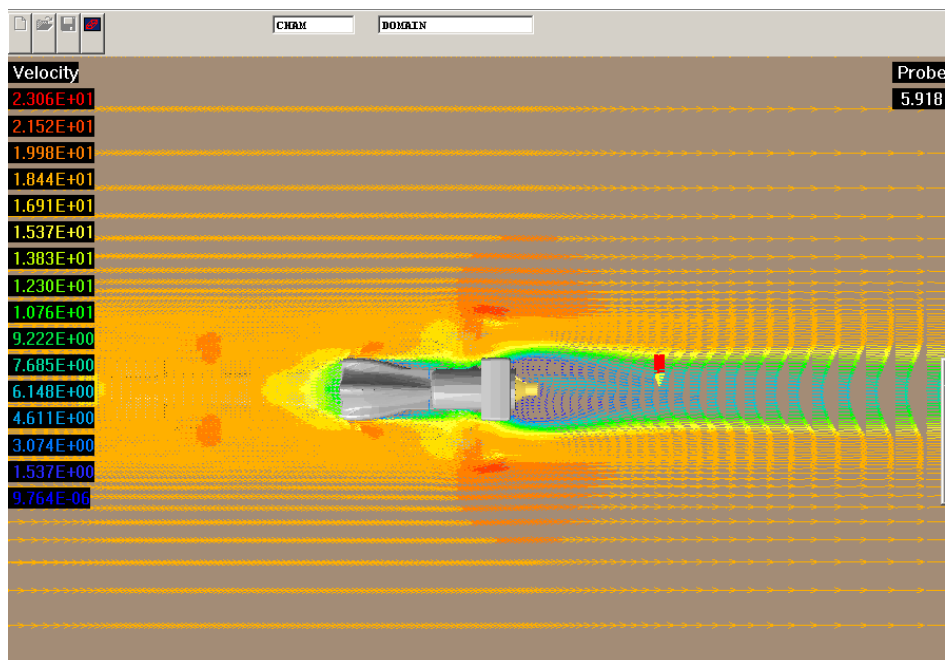
**Note:** 2000 iterations are enough to achieve a reasonable level of convergence; this should take approximately 35 hours using a PIII computer with 500MB of RAM.

### 3. The Results

Plot 2 show the velocity field in the Z-plane. The shape of the car creates two distinct areas of wake. The first is around the front axle with the second occurring behind the car body. The velocity can be seen to increase around the rear wheels with a third set of smaller wakes occurring here.



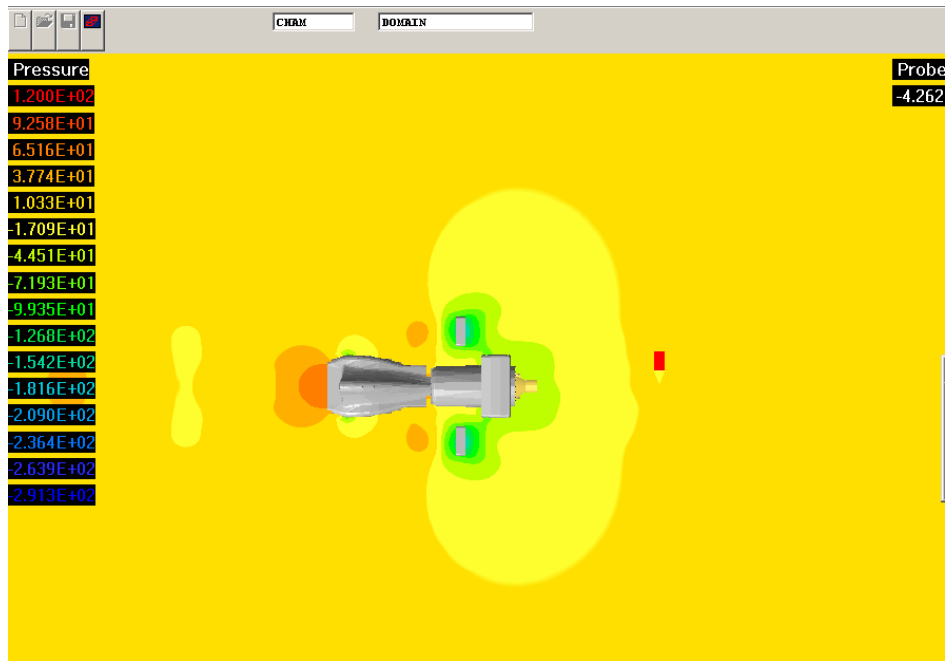
**Plot 2**



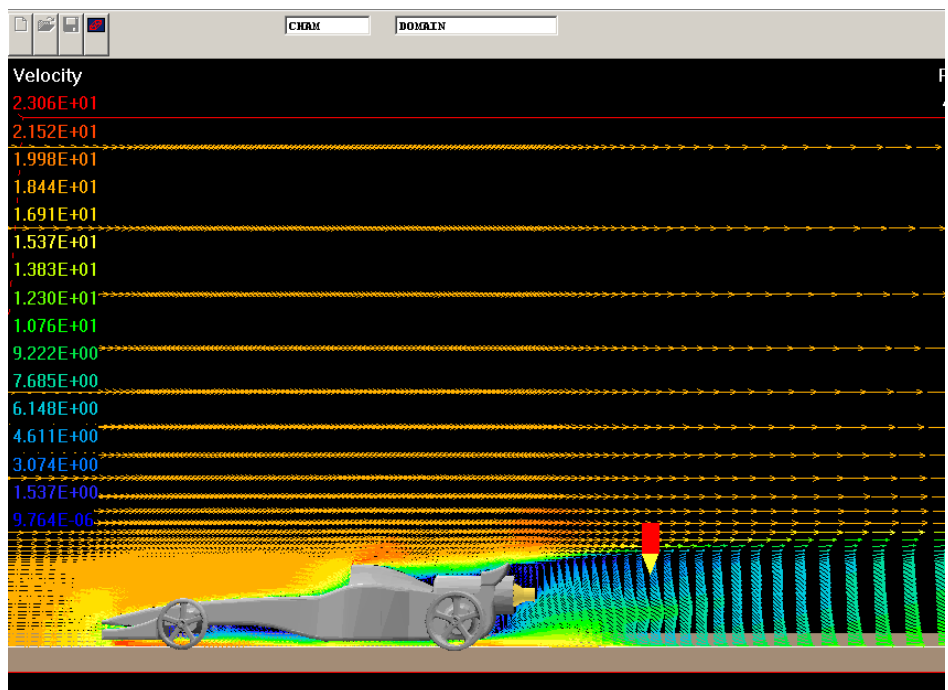
**Plot 3**

Plot 3 shows the velocity profile in the Z axis, but at a slightly higher plane than previous.

Plot 4 below shows the **pressure profile** in the same plane as the previous picture. Notice how the pressure increases just before the raised central section, as the velocity starts to decrease.



**Plot 4**



**Plot 5**

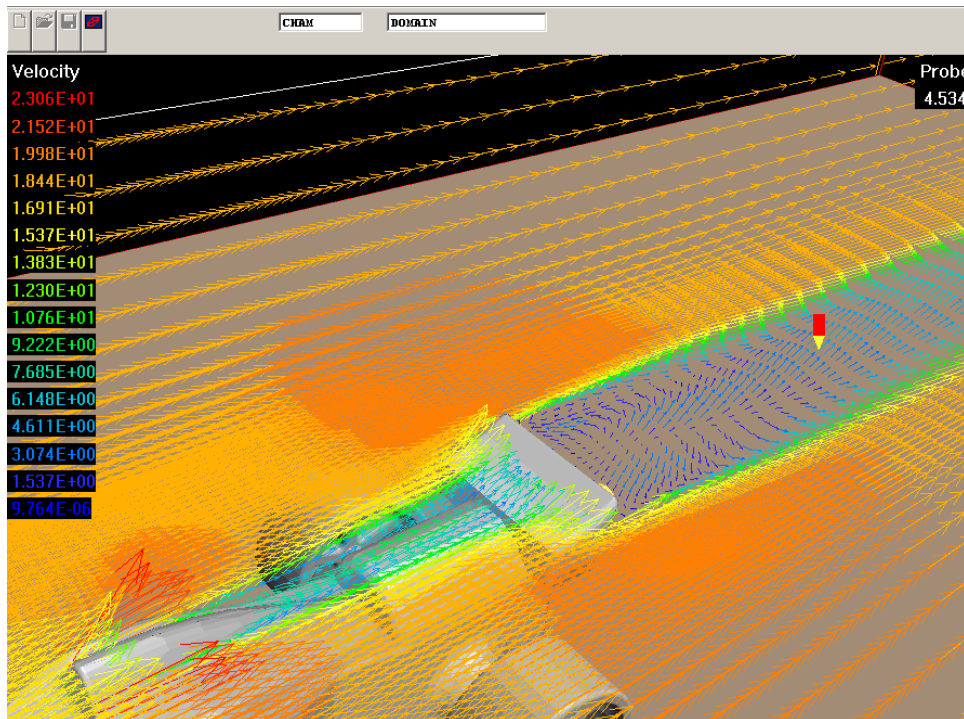
Plot 5 shows the wake left behind the car. Notice the wake left behind both front and rear wings.



Plot 6 below shows the pressure profile along the mid-section of the car in the Y plane.



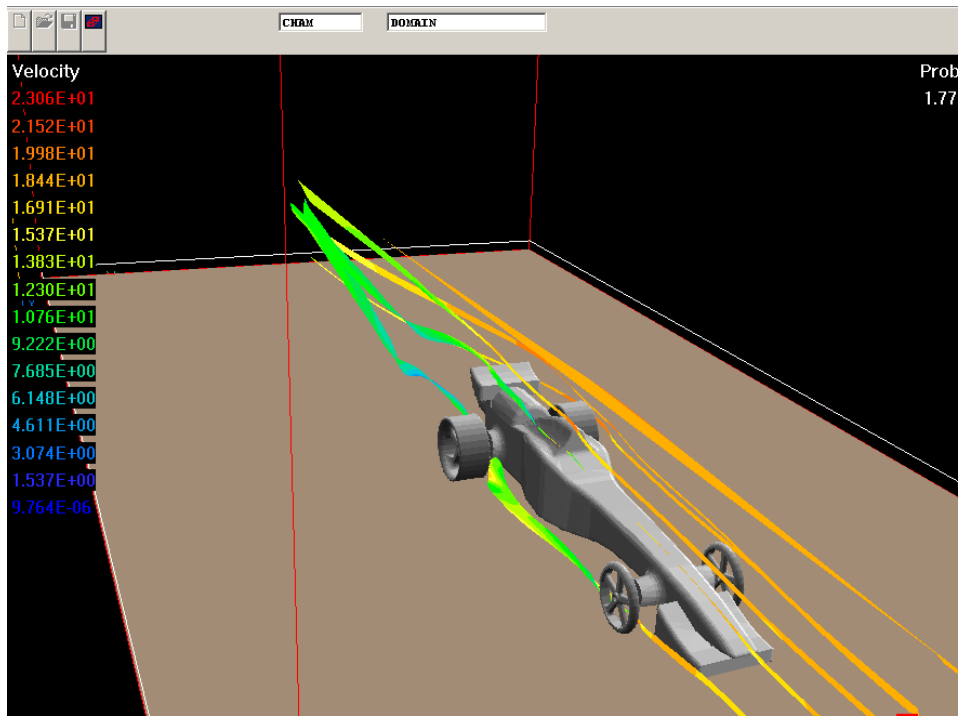
**Plot 6**



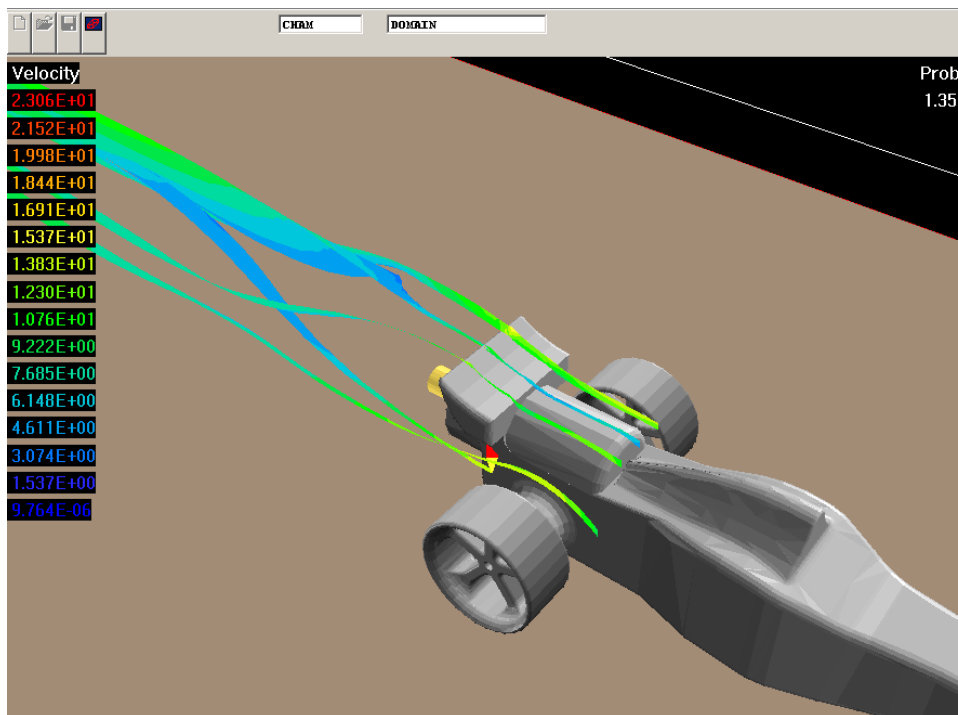
**Plot 7**

The wake trailing from the wing is shown clearly in Plot 7.

The streamline in Plot 8 below shows the general flow around the car.



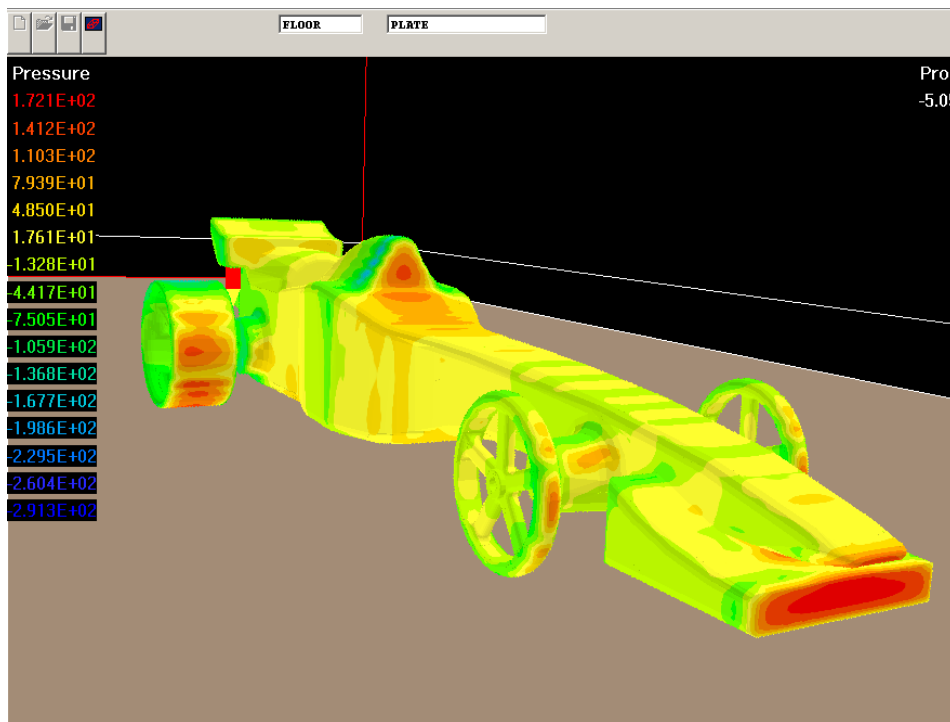
**Plot 8**



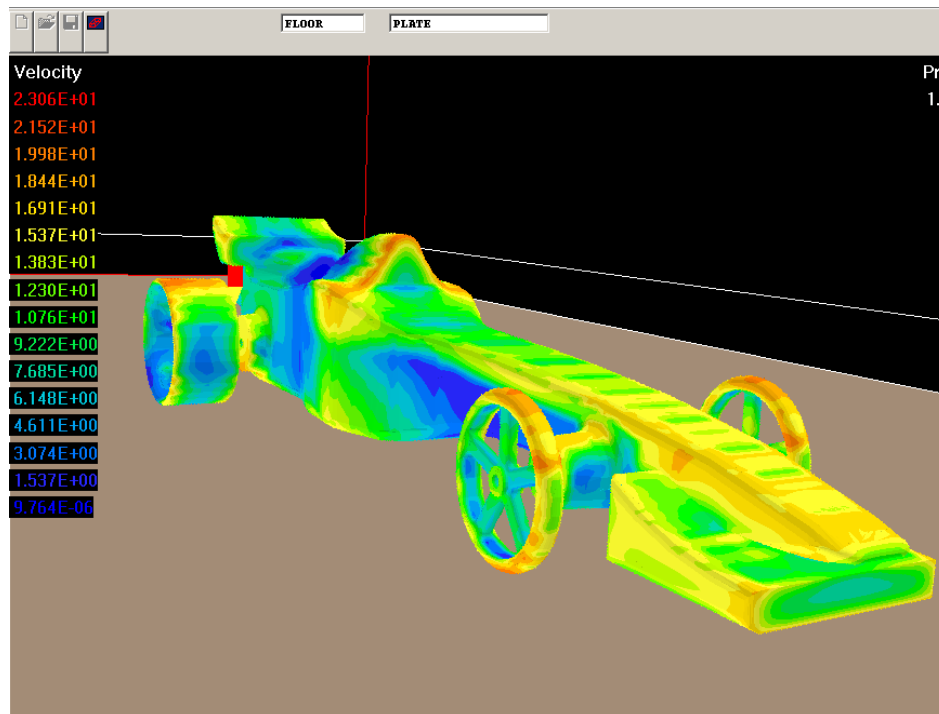
**Plot 9**

Plot 9 shows the streamlines around the rear wing.

Plot 10 shows the pressure profile on the surface of the car. Areas of large surface show increase in pressure, for example the front wing. The equivalent velocity surface plot is shown in Plot 11.



**Plot 10**



**Plot 11**

The areas located in the path of a wake show a zero velocity, for example at the front of the rear wing and the beginning of the side pods.

The drag force data is stored by the **VWT**. The coefficient of drag and the respective forces are given in the table below.

	<b>Car Body</b>	<b>Front Wheel R</b>	<b>Front Wheel L</b>	<b>Rear Wheel R</b>	<b>Rear Wheel L</b>	<b>CO<sub>2</sub> Can</b>
<b>F<sub>x</sub>(N)</b>	-2.8E-01	-2.7E-02	-2.6E-02	-1.4E-01	-1.4E-01	-5.2E-03
<b>F<sub>y</sub>(N)</b>	-3.3E-03	-1.7E-03	3.3E-03	-5.2E-03	1.1E-02	-8.5E-04
<b>F<sub>z</sub>(N)</b>	-9.3E-02	3.9E-04	3.8E-04	7.8E-02	7.9E-02	-1.2E-03
<b>CD<sub>x</sub></b>	3.5E-01	3.3E-01	3.3E-01	3.9E-01	4.0E-01	8.7E-02
<b>CD<sub>y</sub></b>	1.6E-03	1.6E-02	3.2E-02	3.9E-02	8.1E-02	5.1E-03
<b>CD<sub>z</sub></b>	4.2E-02	4.9E-03	4.8E-03	2.3E-01	2.4E-01	6.9E-03

- The main drag components arrive from the **car body** and **rear wheels**.
- The drag force mainly acts in the X direction or **direction of motion**.

#### 4. Conclusion

The flow around the **Jaguar F1** was predicted successfully with velocity, surface contour, pressure, streamline and drag force data being obtained.

The plots have shown the capabilities of the **VWT** whilst gaining an impression of the overall flow regime for this particular geometry.