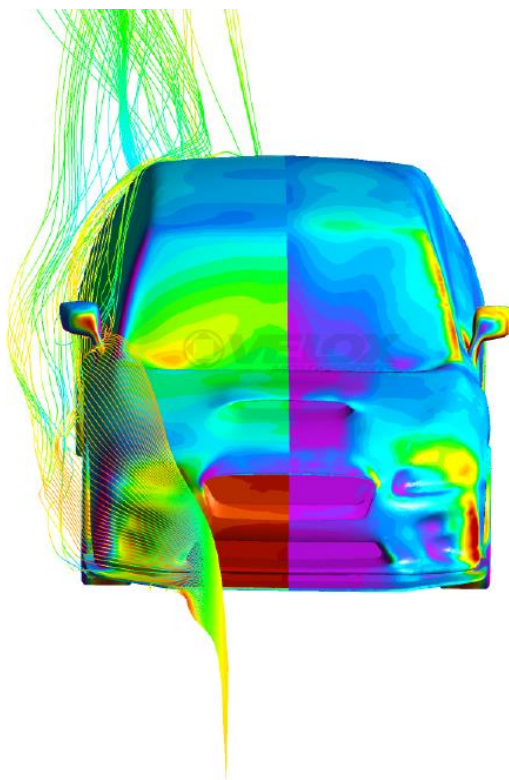




VA Subaru WRX

Performance Gain of Ventus1.5 Package



Author: Paul Lucas

Release Date: 2016/09/21

Approvals: P. Lucas, E. Hazen

Document Revisions

Rev	Date	Author	Description
1	2016/09/21	P. Lucas	Initial Release of Informative Packet
2	2017/08/07	P. Lucas	Company Name Change From Velox to Verus

VA Subaru WRX – Performance Gain of Ventus1 Package

CONTENTS

1.	<u>Executive Summary</u>	3
1.1.	Coefficient Deltas	3
1.2.	Aerodynamic Forces	3
1.3.	Summary Plots	4
2.	<u>ASSUMPTIONS</u>	6
2.1.	Geometry	6
2.2.	Setup	7
3.	<u>MESH QUALITY AND METRICS</u>	7
3.1.	Metrics	7
3.2.	Quality	8
3.3.	Types of Mesh	9
3.4.	Mesh Quality of WRX	9
4.	<u>ANALYSIS SETUP</u>	11
4.1.	General	11
4.2.	Boundary Conditions	12
4.3.	Turbulence Modeling	12
5.	<u>ERRORS IN SIMULATION</u>	23
5.1.	Physical Model	23
5.2.	Discretization Error	23
6.	<u>CONCLUSION</u>	23
7.	<u>REFERENCES</u>	24

VA Subaru WRX – Performance Gain of Ventus1 Package

1. Executive Summary

- Decreased lift
- Drag decreased
- Aerodynamic efficiency improved with the use of the Verus Engineering Ventus1.5 (Front Splitter, Rear Diffuser, Side Splitters) Package

All work was done using ANSYS software: geometry preparation was completed using SpaceClaim, meshing was done with ANSYS Meshing, setup and solution was computed on ANSYS Fluent, and post-processing was completed in CFD-Post.

1.1. Coefficient Deltas

Cd	Cl	Aerodynamic Balance
-0.007	-0.170	43/57

Figure 1.1.1: Coefficient Deltas

1.2. Aerodynamic Forces

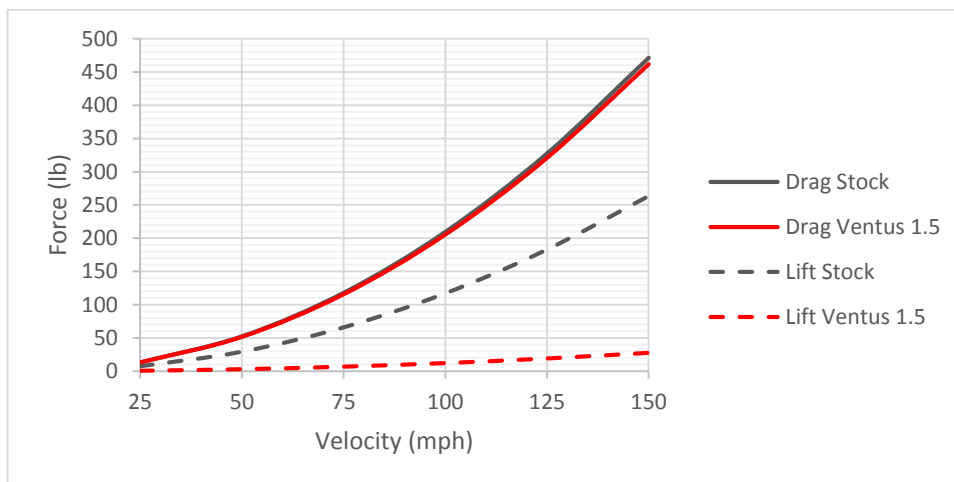


Figure 1.2.1: Aerodynamic Forces at Speed

VA Subaru WRX – Performance Gain of Ventus1 Package

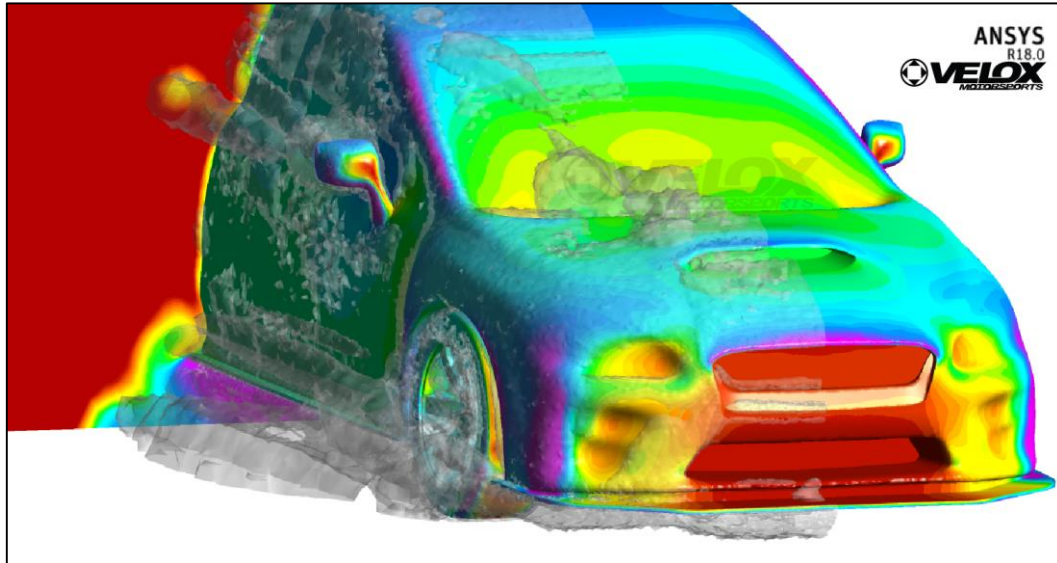
1.3. Summary Plots

Figure 1.3.1: Q-Criterion Plot Front view

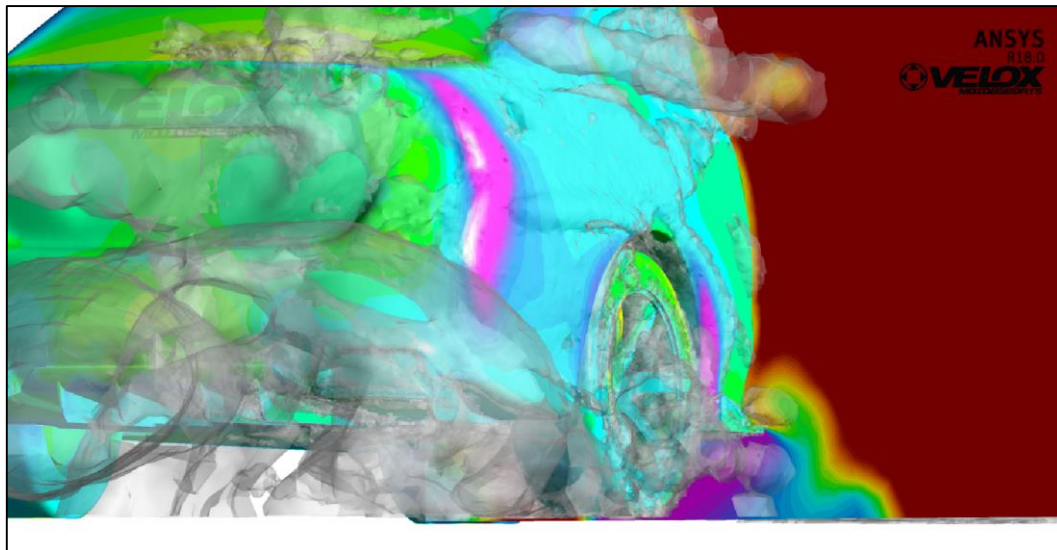


Figure 1.3.2: Q-Criterion Plot Rear View

Confidential: Property of Verus Engineering. Not for Distribution outside intended recipient list.

VA Subaru WRX – Performance Gain of Ventus1 Package

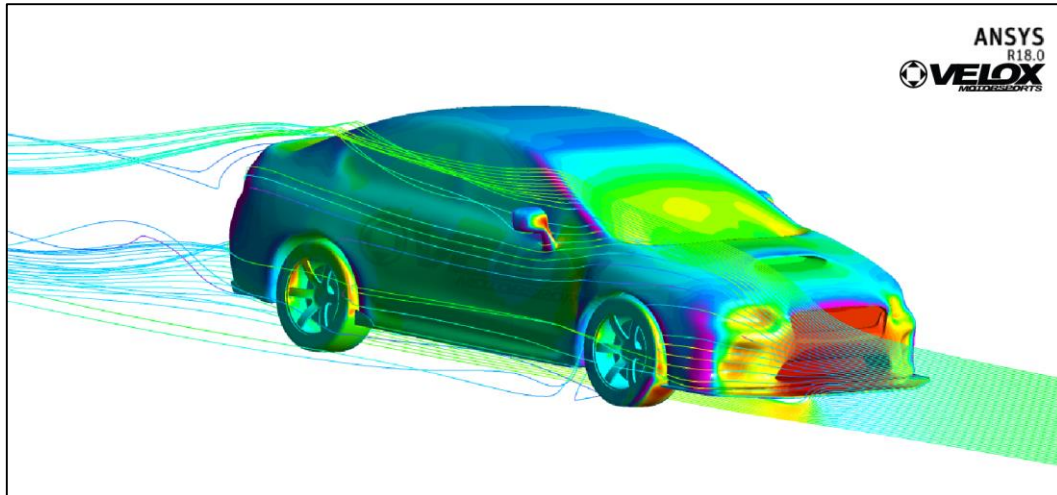


Figure 1.3.3: Stock WRX – Velocity Cut Plane

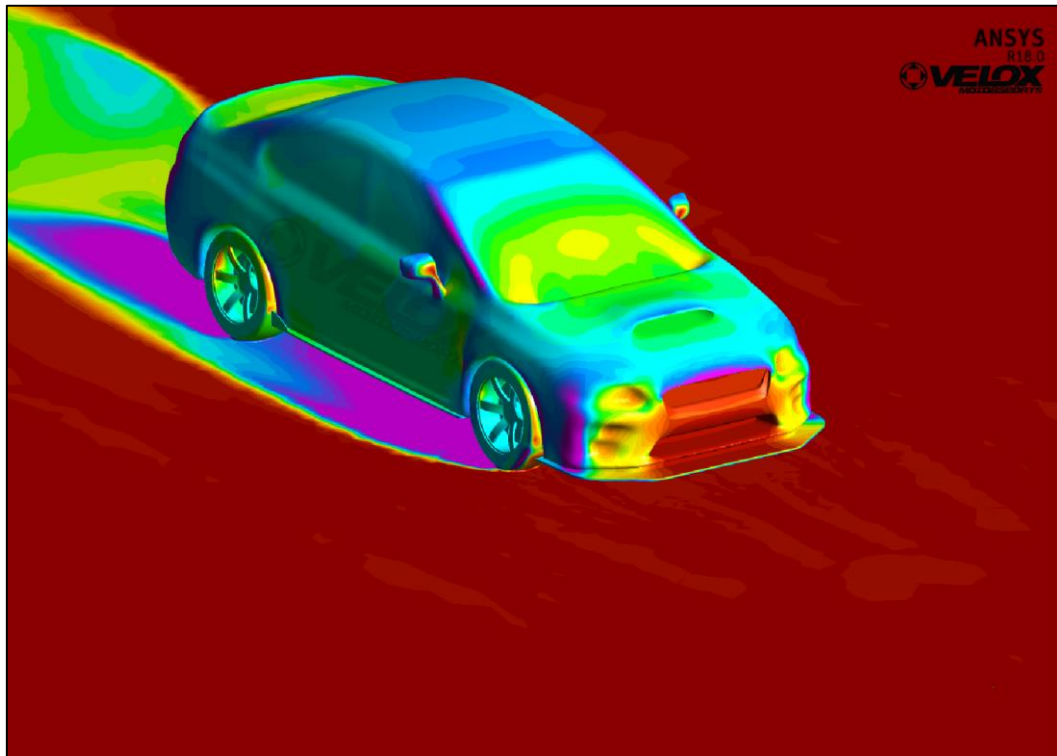


Figure 1.3.4: Ventus1 WRX – Velocity Cut Plane

Notice the vortex coming off of the stake in Figure 1.3.5. Vortices in these locations have a positive impact on diffuser performance. Also notice in Figure 1.3.6 the vortex traveling down the front splitter on the Ventus1 package. Vortex management is a key goal in all aerodynamic development at Verus Engineering.

VA Subaru WRX – Performance Gain of Ventus1 Package

2. ASSUMPTIONS**2.1. Geometry**

In many automotive analysis cases, simplification of the geometry is required. Analyzing a fully detailed automotive case is usually not necessary, and much of the critical information we are after could get lost in the plethora of data. The proper approach to the analysis is to setup a simplified physical model. [1] In the automotive case, we isolate particular aspects of the system to capture and ensure we can set proper boundary conditions. In these cases, we are interested in the overall flow field around the WRX and how the Ventus1.5 package improves the aerodynamic efficiency. The simplifications made were as follows:

- Simplified wheel bodies
- Simplified underbody
- Solid grill surface
- Removed door handles
- Removed antennae
- Removed all car panel gaps

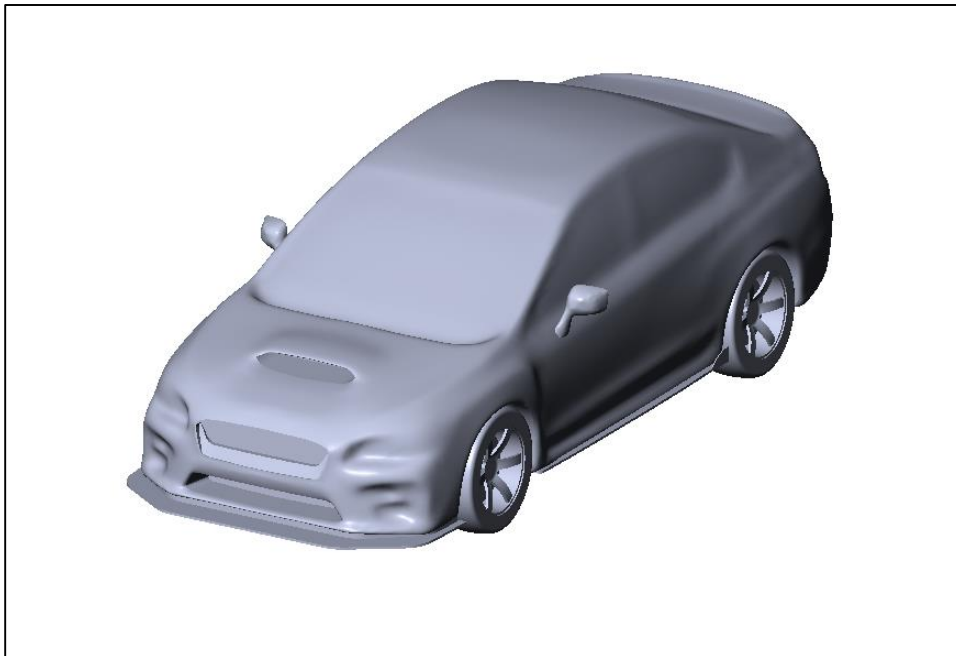


Figure 2.1.2: Geometry Simplifications – Isometric

2.2. Setup

Some physical effects are removed from the analysis because it is not necessary for these cases. We are looking at the overall flow field around the VA WRX, some of the physical models are left out. The simplifications made were as follows:

- No radiator flow
- No under hood flow
- No engine intake
- No exhaust flow

Confidential: Property of Verus Engineering. Not for Distribution outside intended recipient list.

VA Subaru WRX – Performance Gain of Ventus1 Package

3. MESH QUALITY AND METRICS

3.1. Metrics

A proper mesh is essential for a CFD analysis. With CFD, a poor mesh causes the simulation to not converge. Without convergence, the solution will never be reached, and without a good mesh, it is irresponsible to expect accurate results. Here at Verus Engineering we use two main metrics to evaluate mesh quality: skewness and orthogonal quality.

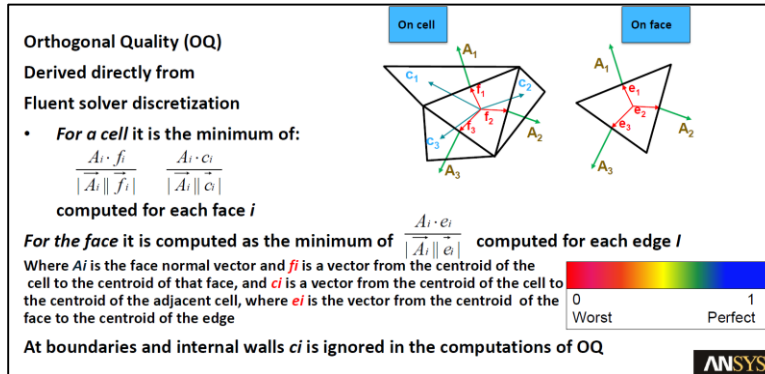


Figure 3.1.1: Orthogonal Quality Metrics

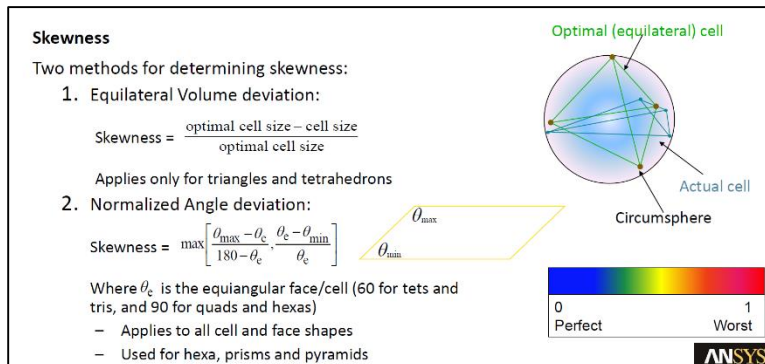


Figure 3.1.2: Skewness Quality Metrics

3.2. Quality

Low orthogonal quality and high skewness values are not recommended for a quality mesh. In general, we always want an orthogonal quality greater than 0.1 and skewness below 0.95. The value may differ depending on the physics and location of the cell.

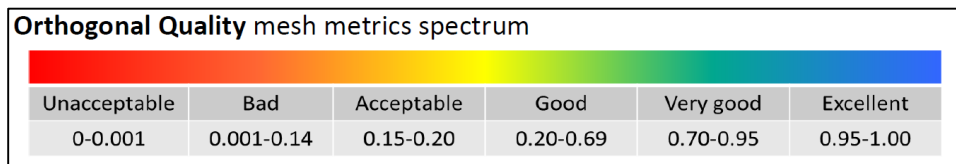


Figure 3.2.1: Orthogonal Quality Metrics

VA Subaru WRX – Performance Gain of Ventus1 Package

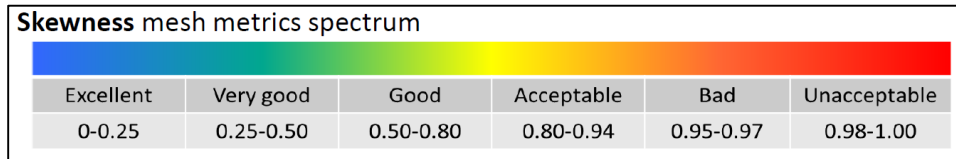


Figure 3.2.2: Skewness Quality Metrics

3.3. Types of Mesh

There are two main types of meshes, or grids: structured and unstructured. Structured grids generally have better convergence and higher accuracy with a more simple flow field. Unstructured grids are more commonly used for Verus Engineering applications since geometry and flow fields are complex.

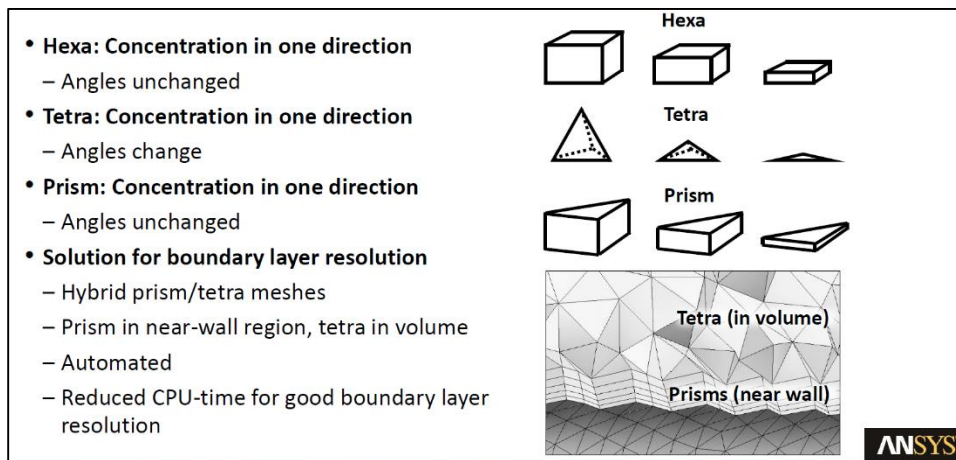


Figure 3.3.1: Hexahedral versus Tetrahedral

An unstructured mesh using prism layer for boundary layer resolution and tetrahedral in the volume was utilized for the VA WRX.

3.4. Mesh Quality of WRX

All meshing is completed using ANSYS Meshing. Verus Engineering uses ANSYS Meshing to combine the strengths of ICFM CFD, TGRID, CFX-Mesh, and Gambit. ANSYS Meshing allows Verus Engineering to setup a quality grid for proper solution and convergence.

VA Subaru WRX – Performance Gain of Ventus1 Package

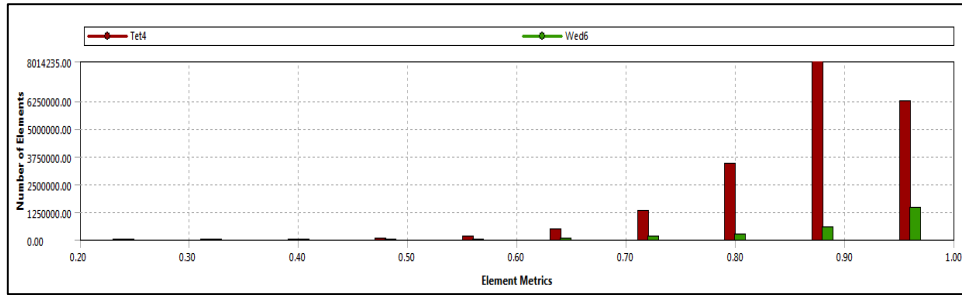


Figure 3.4.1: Mesh Quality of WRX

Referring to Figure 2.4.1, orthogonal quality did not drop below 0.18. This is in the acceptable zone for the minimum orthogonal quality. The majority of the mesh is in the very good to excellent range.

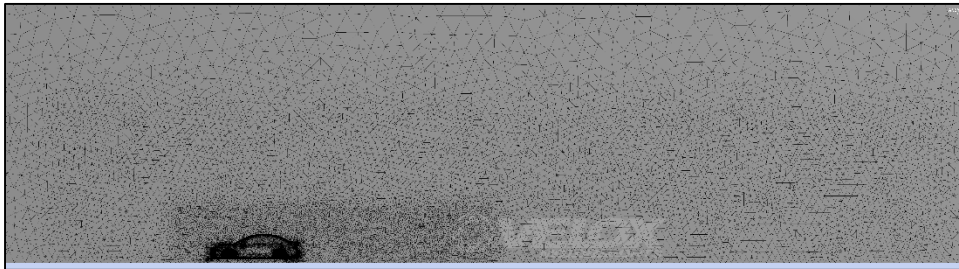


Figure 2.4.2: Mesh Wake Zone Refinement

4. ANALYSIS SETUP

4.1. General

ANSYS Fluent – a finite volume discretization scheme - is used for solving all of Verus Engineering external aerodynamics cases. The solver used is the semi-implicit method for pressure-linked equations, SIMPLE. [2] These cases were solved using the steady-state method which ignores the higher order terms dealing with time.

4.2. Boundary Conditions

Boundary conditions are used to define the computational problem. Without boundaries, nothing can be solved, and proper boundary conditions are a must for an accurate analysis. A full understanding of the physics of the problem is a must. CFD only emulates the condition of the road and track, it does not reproduce it. Simulations inherently deviate from reality and it is often hard to quantify all the sources of error. The cases were solved using symmetry on the XZ Plane. This decreased the mesh the mesh quantity by cutting the computational domain in half, thus decreases the simulation time to convergence. Using symmetry on the XZ Plane is standard in the industry unless you are modeling yaw. The case is modeled using a virtual wind tunnel which is discretized and used as the fluid volume. The air then flows over the stationary car in the boundary. The ground and wind tunnel walls have the shear equal to zero to not have boundary layer growth which simulates the rolling road condition to have a proper flow regime. The wheels are setup as a rotational moving wall boundary.

VA Subaru WRX – Performance Gain of Ventus1 Package

Inlet	Inlet Condition	45 m/s
Outlet	Pressure Outlet Condition	0 Pa (Gauge)
Symmetry Plane	Symmetry	N/A
Ground	Wall Condition	Shear = 0
Wind Tunnel Walls	Wall Condition	Shear = 0
WRX	Wall Condition	No Slip Shear
Wheels	Wall Condition	144 rad/s

Figure 3.2.1: Boundary Conditions

4.3. Turbulence Modeling

The majority of fluid flows in engineering cases are turbulent. Because of this, a proper turbulence model is needed for proper results and flow regimes. The k-omega SST turbulence model was used in these cases. The k-omega is great for solving flows in the viscous sublayer (boundary layer). The SST allows the model to solve free-stream area where the normal k-omega was too sensitive to free-stream turbulences. [3]

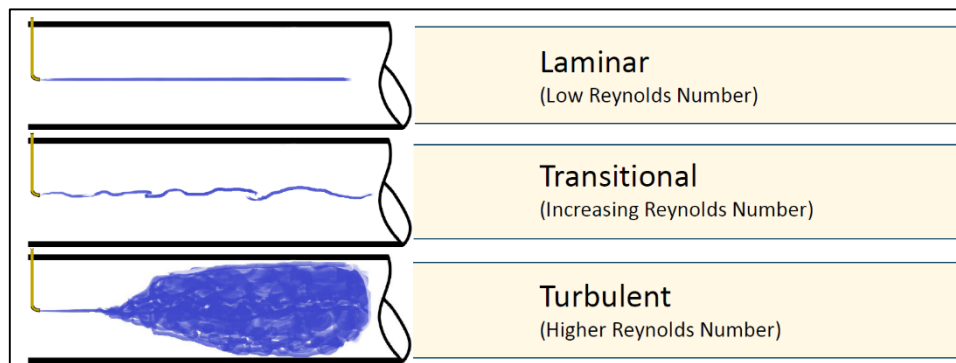


Figure 3.3.1: Flow Classifications

5. ERRORS IN SIMULATION

5.1. Physical Model

Some of the errors in the physical model was discussed in Geometry Assumptions. The substitution of a CAD model from a real physical model in an inevitable source of error. The CAD model only also represents part of the physical system, others are either represented with boundary conditions or left out of the analysis.

5.2. Discretization Error

Discretization errors are those errors that occur from the representation of the governing flow equations and other physical models as algebraic expressions in a discrete domain of space and time. As the mesh is refined, the solution should become less sensitive to discretization error. These errors can be analyzed using grid convergence study. [4]

6. CONCLUSION

Confidential: Property of Verus Engineering. Not for Distribution outside intended recipient list.

VA Subaru WRX – Performance Gain of Ventus1 Package

The Verus Engineering Ventus1 package decreased the overall lift and reduced drag of the VA WRX. Since the diffuser and splitter decreased lift and reduced drag, the overall aerodynamic efficiency was improved. The diffuser helped create vortices that kept flow attached to the diffuser surface. The diffuser created a low pressure zone in the rear while the splitter created a low pressure zone in the front. This equated to a well-proportioned aerodynamic balance. The Verus Engineering Ventus1 package helps the VA WRX get more from the tires at the track and also increases gas mileage on a daily driver. The Ventus1 package will be great on either a normal daily driver or a track queen.

VA Subaru WRX – Performance Gain of Ventus1 Package7. REFERENCES

- [1] Zikanov, Oleg. *Essential Computational Fluid Dynamics*. 1st Edition. Joh Wiley & Sons, Inc., 2010.
- [2] Patankar, Suhas. *Numerical Heat Transfer and Fluid Flow*. 1st Edition. Hemisphere Publishing Corporation, 1980.
- [3] Menter, Florian. Improved Two-Equation k-w Turbulence Models for Aerodynamic Flows. NASA Technical Memorandum, no. 103975 (1992).
- [4] Slater, John. 2008. Uncertainty and Error in CFD Simulations.
<https://www.grc.nasa.gov/WWW/wind/valid/tutorial/errors.html>
(Acessed 2016-09-20).