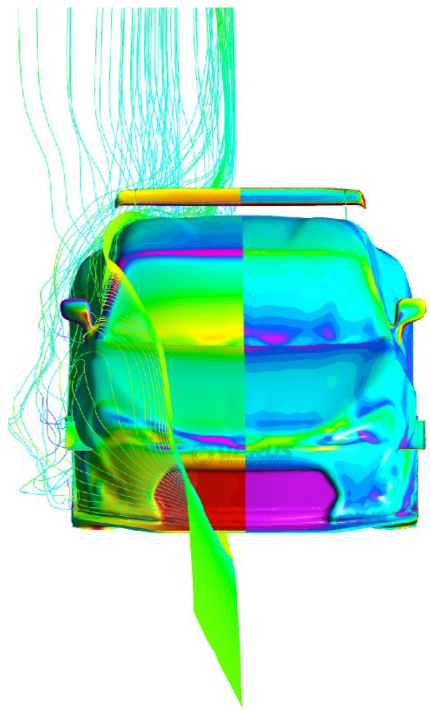
FRS BRZ GT86

Performance of Verus Engineering Ventus 2 Package



Author: P. Lucas
Release Date: 2017/08/09
Approvals: E. Hazen

Document Revisions

Rev	Date	Author	Description
01	2017/08/09	P. Lucas	Issued for Release

Verus Engineering
Indianapolis, IN
sales@verus-engineering.com

CONTENTS

FRS BRZ GT86 – Performance of Verus Engineering Ventus 2 Package

1. <u>Executive Summary</u>
   - Increased downforce
   - Increased drag
   - Improved overall aerodynamic efficiency
   - Ventus 2 Package is rear diffuser, rear spats, front splitter, side splitter, front endplates, dive planes, and rear wing

All work was done using ANSYS software: geometry preparation was completed using SpaceClaim, meshing was done with Fluent 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 (Fr/Rr)
+0.11	-0.403	40/60

Figure 1.1.1: Coefficient Deltas

1.2. Aerodynamic Forces

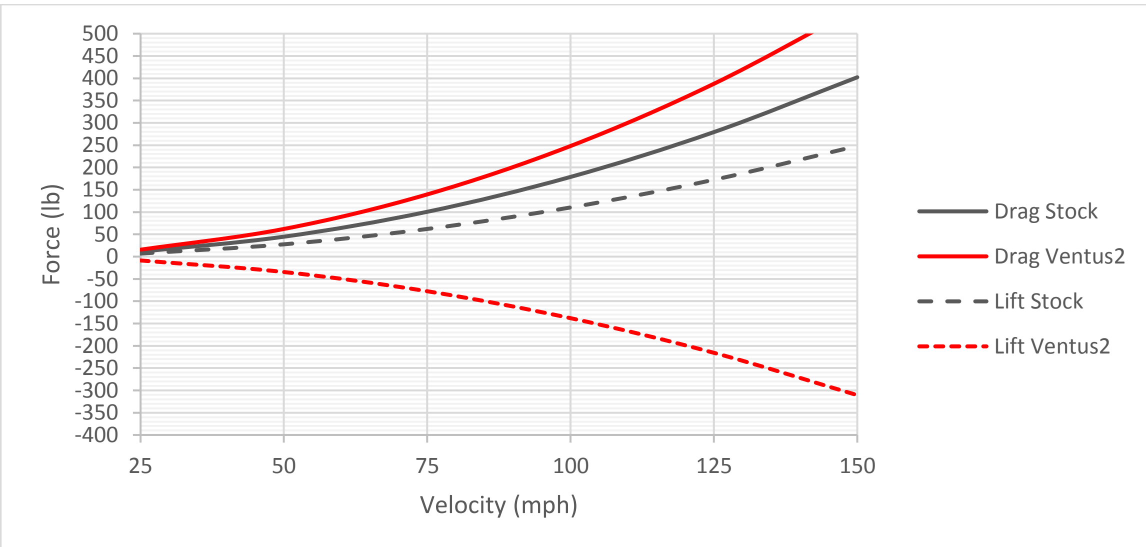


Figure 1.2.1: Aerodynamic Forces at Speed

Aerodynamic forces change with the square of velocity. This is why a single downforce number cannot be given unless it is at a specific speed. Most OEM road vehicles create lift from the factory. With the Ventus 2 package, downforce is increased quite substantially. An increase in overall aerodynamic efficiency will lead to faster track times around the track.

1.3. Summary Plots

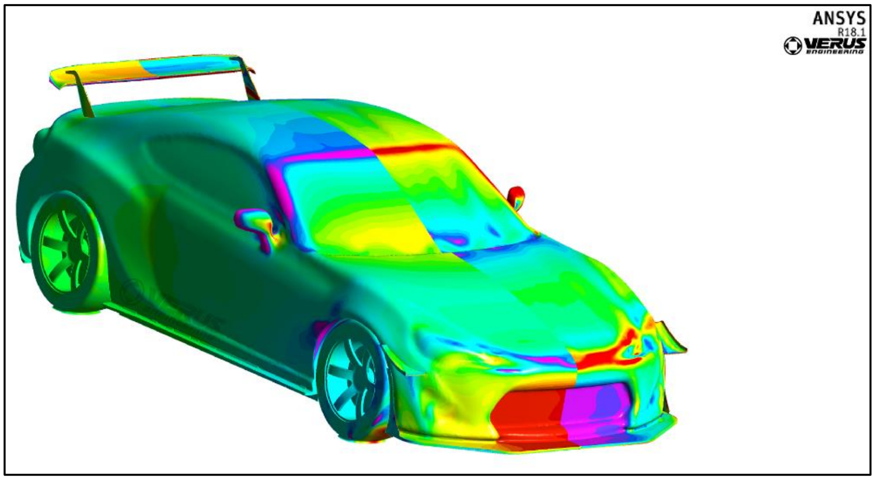


Figure 1.3.1: Cp Plot of the FRS BRZ GT86

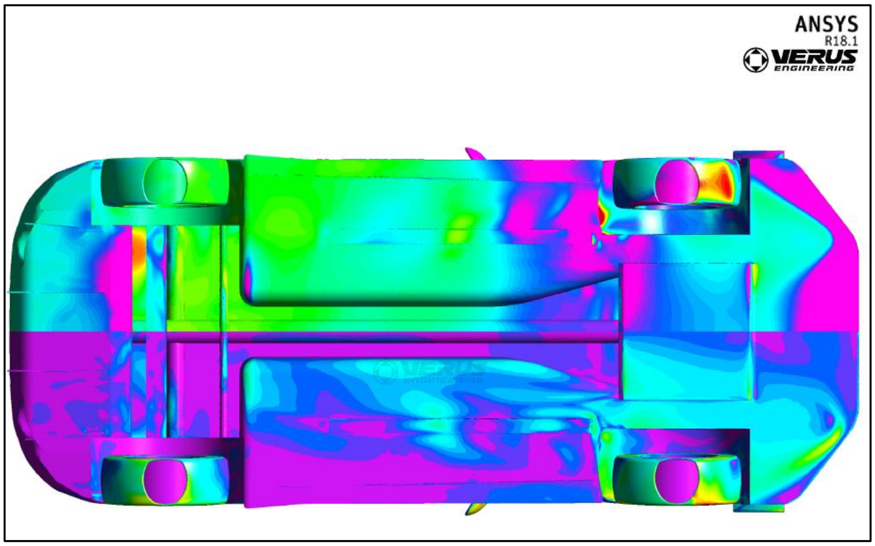


Figure 1.3.2: Cp Plot (Top) of the Diffuser and Splitter

Figures 1.3.1 and 1.3.2, we have a Cp Plot (Coefficient of Pressure) which quickly and easily shows us the pressure on different surfaces of the car including the Verus Engineering components. Low pressure is wanted on the bottom side of the components like the front splitter, diffuser, side splitter, wing, and dive planes. Figure 1.3.2 specifically shows the strong low pressure on the underside of the diffuser and splitter.

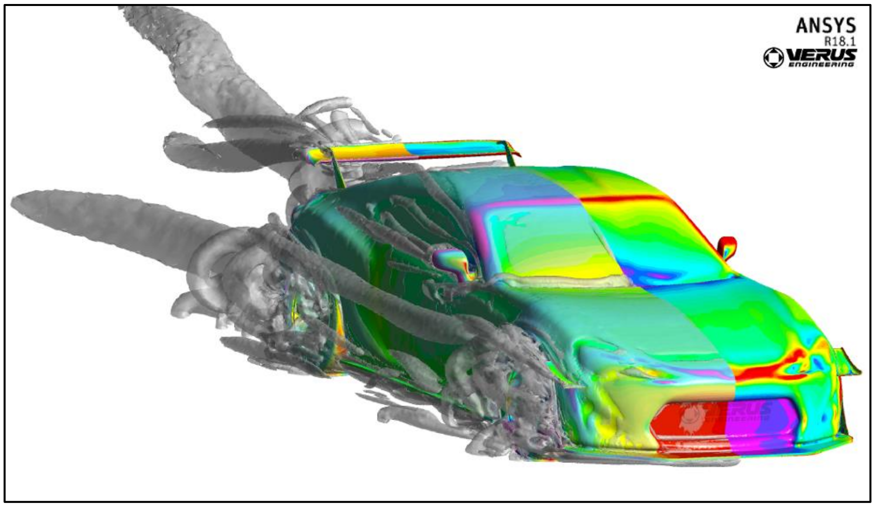


Figure 1.3.3: Q-Criterion Front

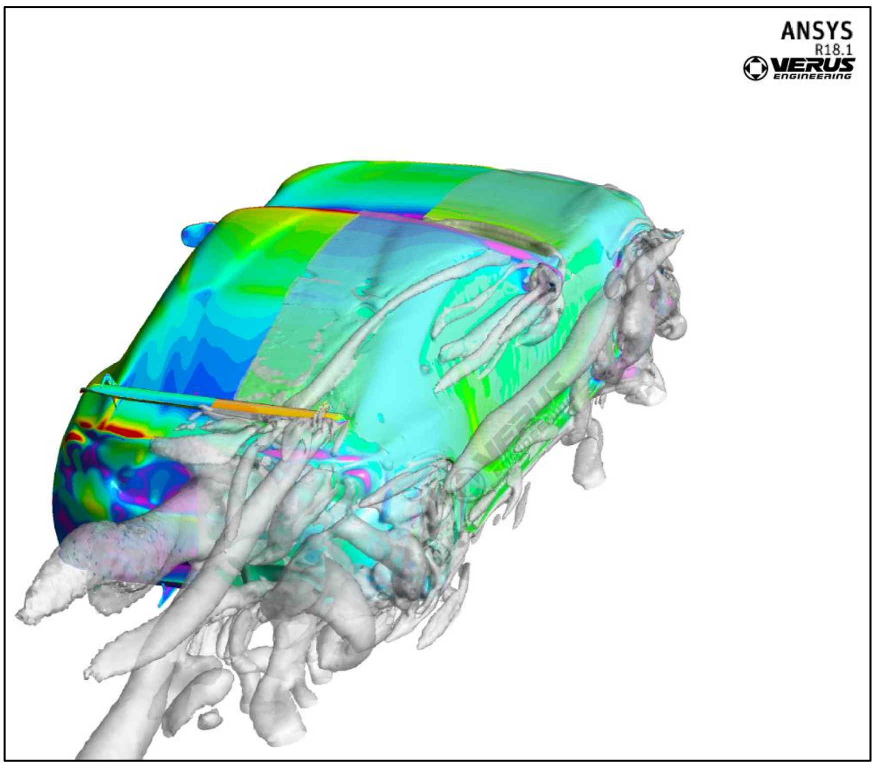


Figure 1.3.4: Q-Criterion Rear

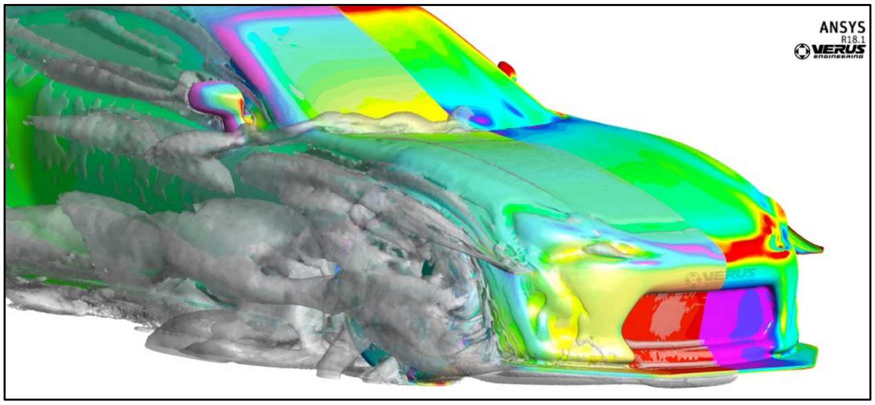


Figure 1.3.5: Q-Criterion Close Up on Front

In Figures 1.3.3, 1.3.4, and 1.3.5 show Q-Criterion plots which highlight areas of vorticity. Q-Criterion helps locate vortices and allows us to direct vortices to locations where they can be utilized efficiently and positively. Notice the strong vortices forming from the dive plane and the rear wing. This strong vortex on the dive plane is one of the reasons flow can stay attached at such a high angle of attack. The strong vortex off the rear wing is caused by the strong pressure gradients between the top and bottom of the wings surface.

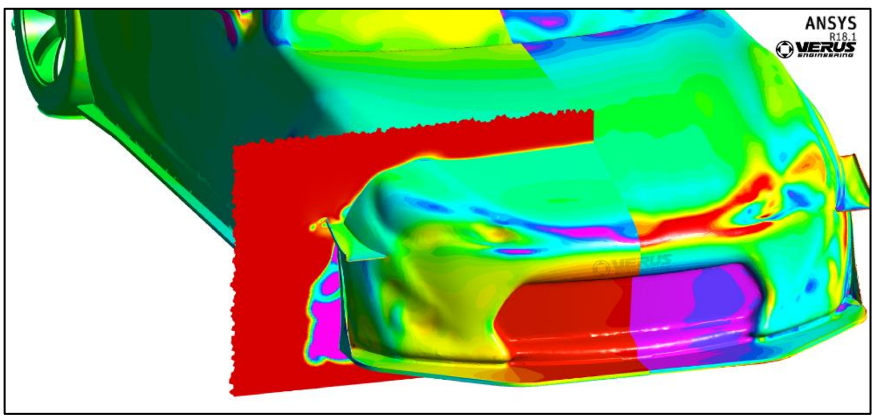


Figure 1.3.6: CpT Plot around Dive Plane

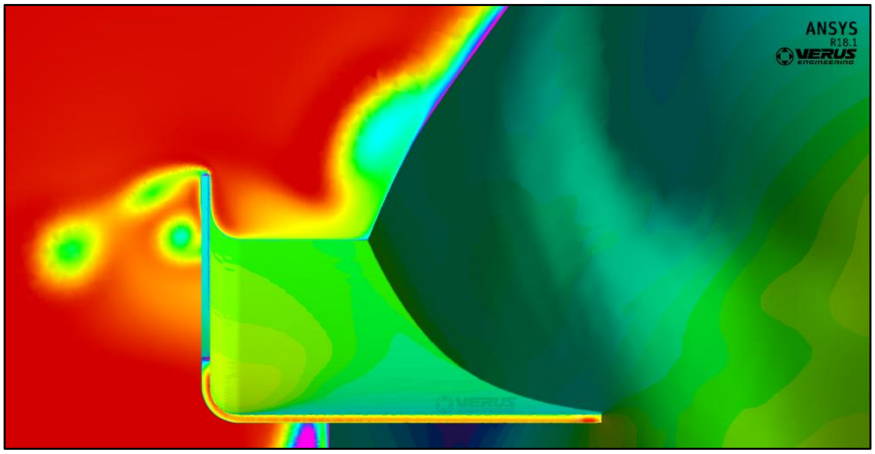


Figure 1.3.7: CpT Plot Close Up at Dive Plane

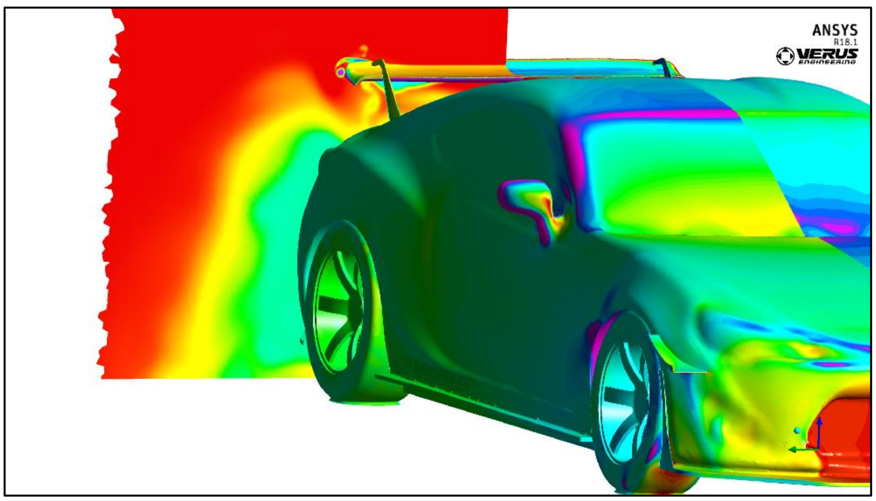


Figure 1.3.8: CpT Plot Just Behind the Rear Wing

In Figures 1.3.6, 1.3.7, and 1.3.8, we have a CpT (coefficient of total pressure) plot. CpT locates free stream flow, red signifying free stream flow. The other colors represent disruptions in this flow. CpT plots also depict vortices. In Figure 1.3.7, notice the vortices forming on the dive plane. These vortices help keep flow attached at high angles of attack. The high strength vortex coming off the end of the wing can be seen in Figure 1.3.8. This vortex is caused by the pressure gradient between the top and bottom surface.

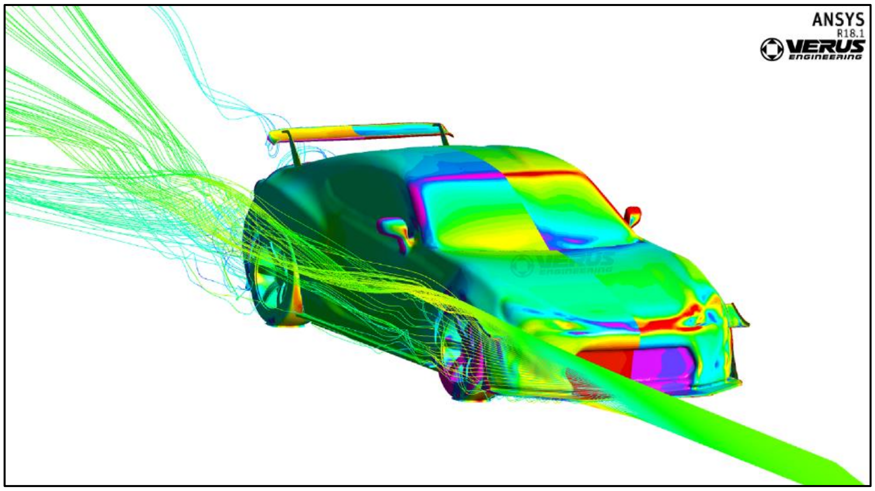


Figure 1.3.9: Front ISO Streamlines

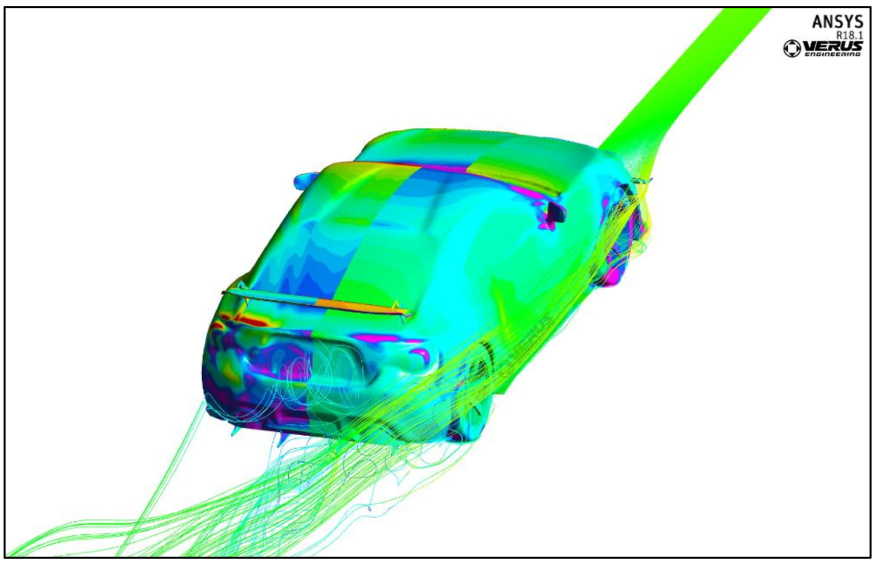


Figure 1.3.10: Rear ISO Streamlines

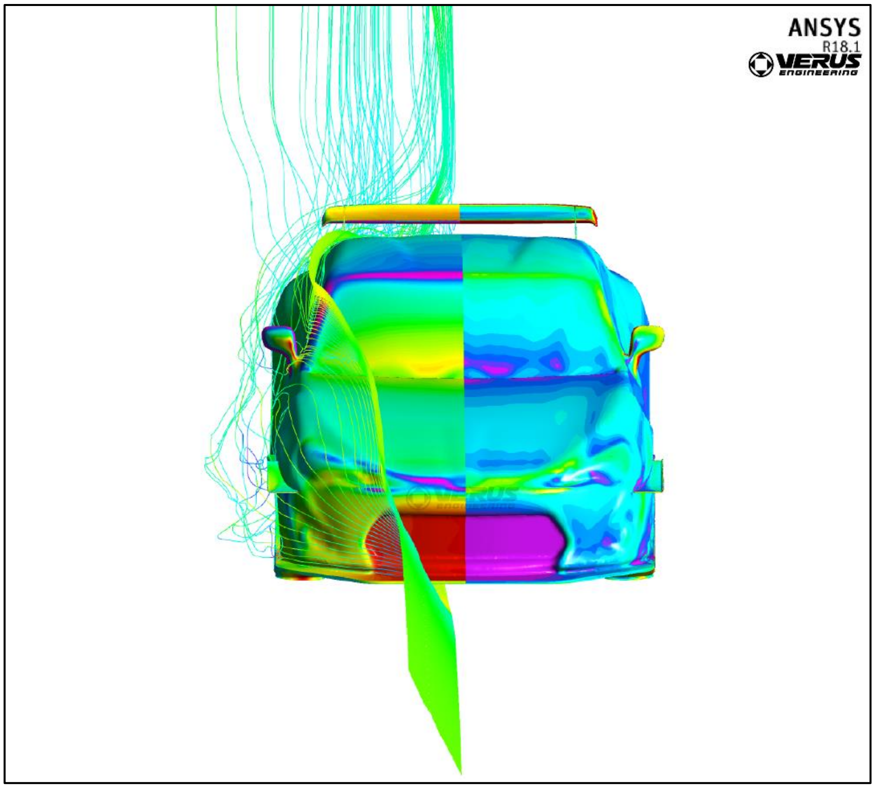


Figure 1.3.11: Front Streamlines

Figures 1.3.9, 1.3.10, and 1.3.11 show streamlines over cars body. Streamlines are another tool used to locate vortices and visualize the general flow field around the vehicle. Streamlines are used by us to verify devices are utilized and directing airflow correctly. Figure 1.3.9 specifically shows the vortices coming off the dive plane.

2. <u>ASSUMPTIONS</u>

2.1. Geometry

In many automotive analysis cases, simplification of the geometry is required. Analyzing a fully detailed automotive case is 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 FRS BRZ GT86 and how the Verus Engineering Ventus 2 Package improves the aerodynamic efficiency. The simplifications made were as follows:

- Simplified wheels and brakes
- Simplified underbody
- Simplified suspension
- Solid grill surface
- Removed door handles
- Removed all car panel gaps

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 FRS BRZ GT86, 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

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.

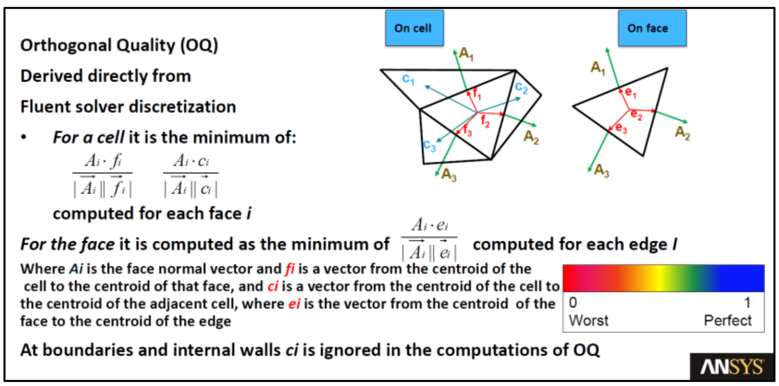


Figure 3.1.1: Orthogonal Quality Metrics

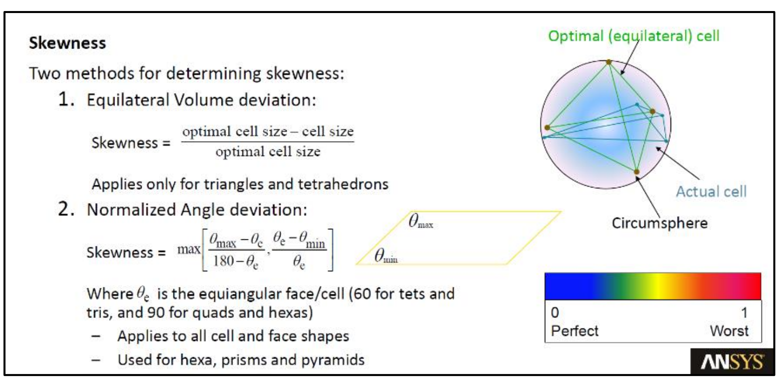


Figure 3.1.2: Skewness Quality Metrics

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.

Orthogonal Quality mesh metrics spectrum

Unacceptable	Bad	Acceptable	Good	Very good	Excellent
0-0.001	0.001-0.14	0.15-0.20	0.20-0.69	0.70-0.95	0.95-1.00

Figure 3.2.1: Orthogonal Quality Metrics

Skewness mesh metrics spectrum

Excellent	Very good	Good	Acceptable	Bad	Unacceptable
0-0.25	0.25-0.50	0.50-0.80	0.80-0.94	0.95-0.97	0.98-1.00

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.

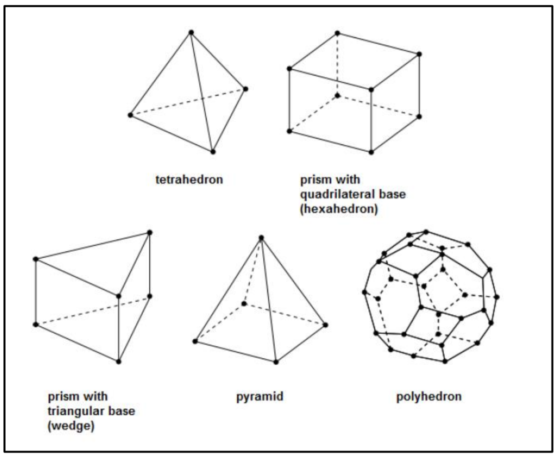


Figure 3.3.1: Volume Cell Shapes

An unstructured mesh using prism layer for boundary layer resolution and polyhedron for the volume was utilized in this case. An advantage that polyhedral meshes have shown relative to tetrahedral or hybrid meshes is the lower overall cell count, almost 3-5 times lower than the original cell count. [2] Also since cell count is lower per the equal refinement, convergence will generally be faster.

3.4. Mesh Quality

All meshing is completed using Fluent Meshing. Fluent Meshing allows Verus Engineering to setup a quality grid for proper solution and convergence.

Orthogonal quality did not drop below 0.21. This is in the acceptable zone for the minimum orthogonal quality. The majority of the mesh is in the very good to excellent range. The volume mesh had a cell count just over 17 million.

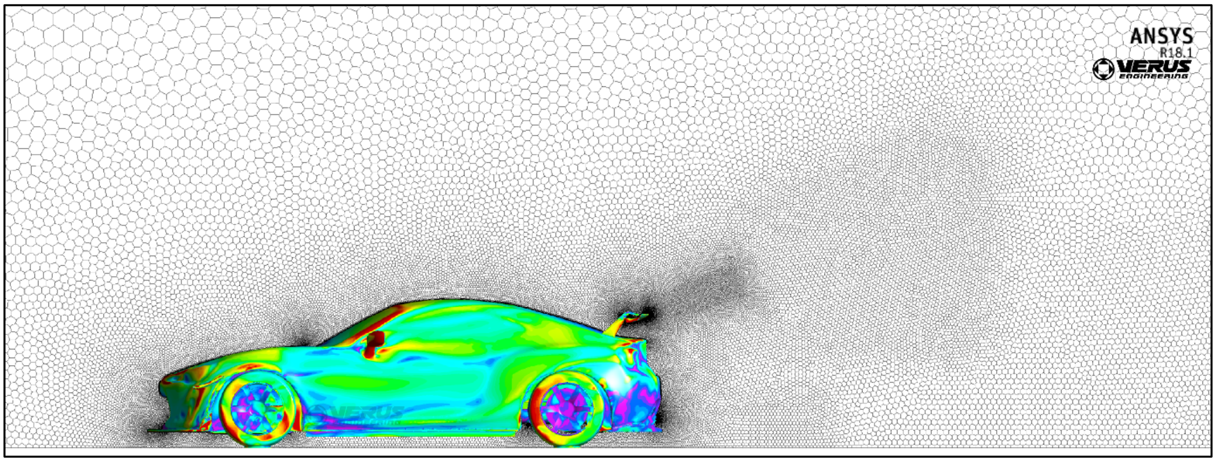


Figure 3.4.1: Mesh Domain

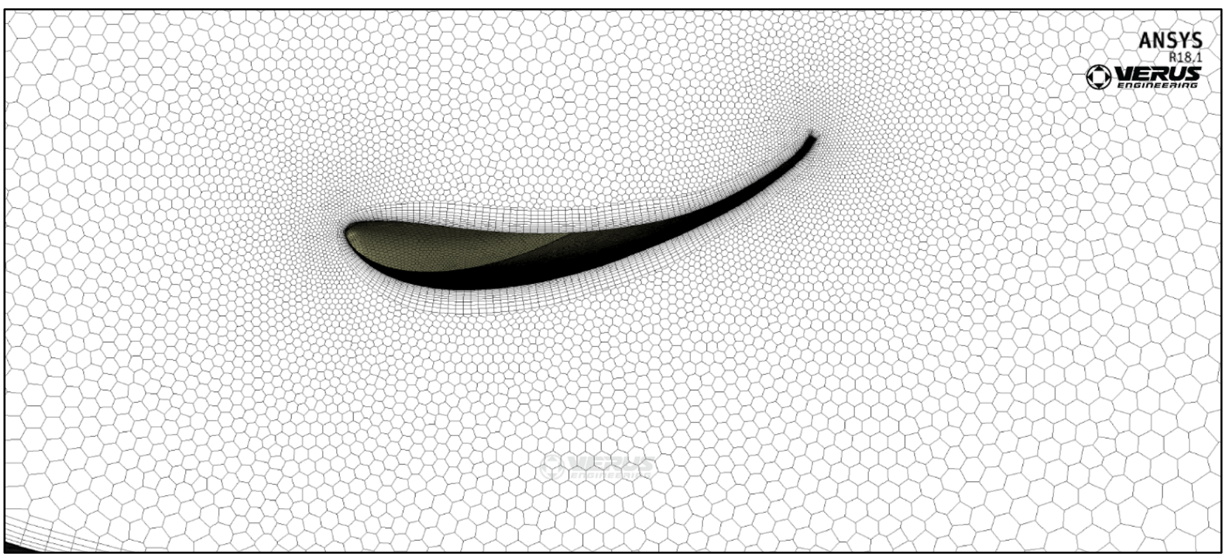


Figure 3.4.2: Prism Layers on Wing

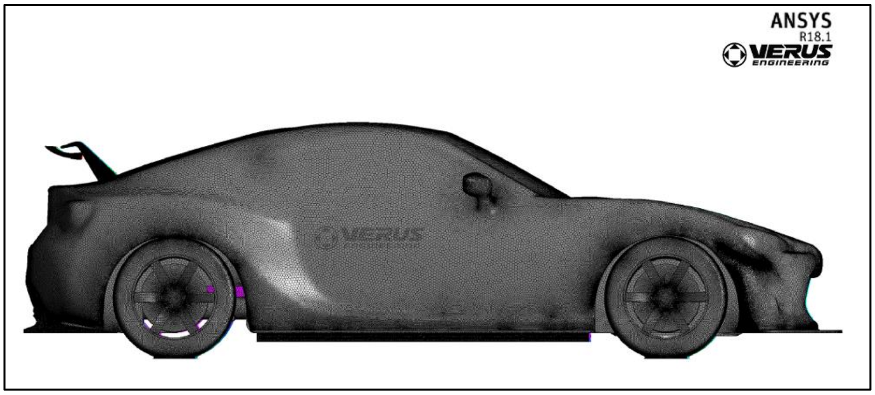


Figure 3.4.3: Surface Mesh

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. [3] 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 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.

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	Symmetry	N/A
Car	Wall Condition	No Slip Shear
Wheels	Wall Condition	141 rad/s
Wing	Wall Condition	No Slip Shear

Figure 4.2.1: 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. [4]

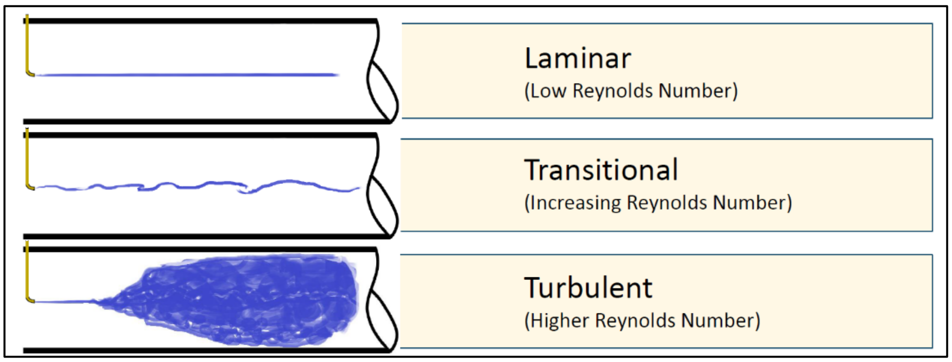


Figure 4.3.1: Flow Classifications

5. ERRORS IN SIMULATION

5.1. Physical Model

Some of the errors in the physical model were discussed in Geometry Assumptions. The substitution of a CAD model from a real physical model is an inevitable source of error. The CAD model only represents part of the physical system, others are either represented with boundary conditions or removed from 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. [5]

6. CONCLUSION

The goal of the Ventus 2 package for the FRS BRZ GT86 is to improve aerodynamic performance over stock. The Ventus 2 package is designed for the enthusiast who frequent track days and want an effective aerodynamic package that balances the car at speed. All the components were designed to work together as a package.

REFERENCES

[1] Zikanov, Oleg. *Essential Computational Fluid Dynamics*. 1st Edition. John Wiley & Sons, Inc., 2010.
[2] ANSYS Fluent Meshing User's Guide, 18.1, April 2017.
[3] Patankar, Suhas. Numerical Heat Transfer and Fluid Flow. 1st Edition. Hemisphere Publishing Corporation, 1980.
[4] Menter, Florian. Improved Two-Equation k-w Turbulence Models for Aerodynamic Flows. NASA Technical Memorandum, no. 103975 (1992).
[5] Slater, John. 2008. Uncertainty and Error in CFD Simulations. https://www.grc.nasa.gov/WWW/wind/valid/tutorial/errors.html (Accessed 2016-09-20).