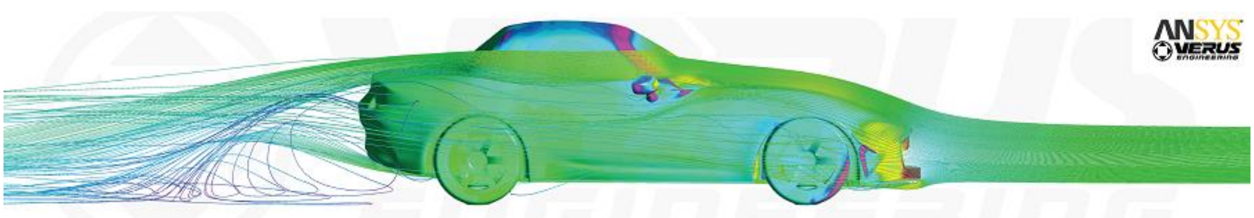
ND Mazda Miata

Performance of Verus Engineering Dive Planes



Author: E. Hazen
Release Date: 17/12/03
Approvals: P. Lucas

Document Revisions

Rev	Date	Author	Description
01	17/12/03	E. Hazen	Issued for Release

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

CONTENTS

Dive Plane Analysis on the ND Mazda Miata

1. Executive Summary
   - Reduce front end lift
   - Reduce vehicle drag
   - Shift aero balance forward

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.

The analysis was performed with our rear diffuser and street front splitter.

1.1. Coefficient Deltas

ΔCd	ΔCl	Aerodynamic Balance (Fr/Rr)
-0.0303	-0.0974	44/56

Figure 1.1.1: Coefficient Deltas

1.2. Aerodynamic Forces

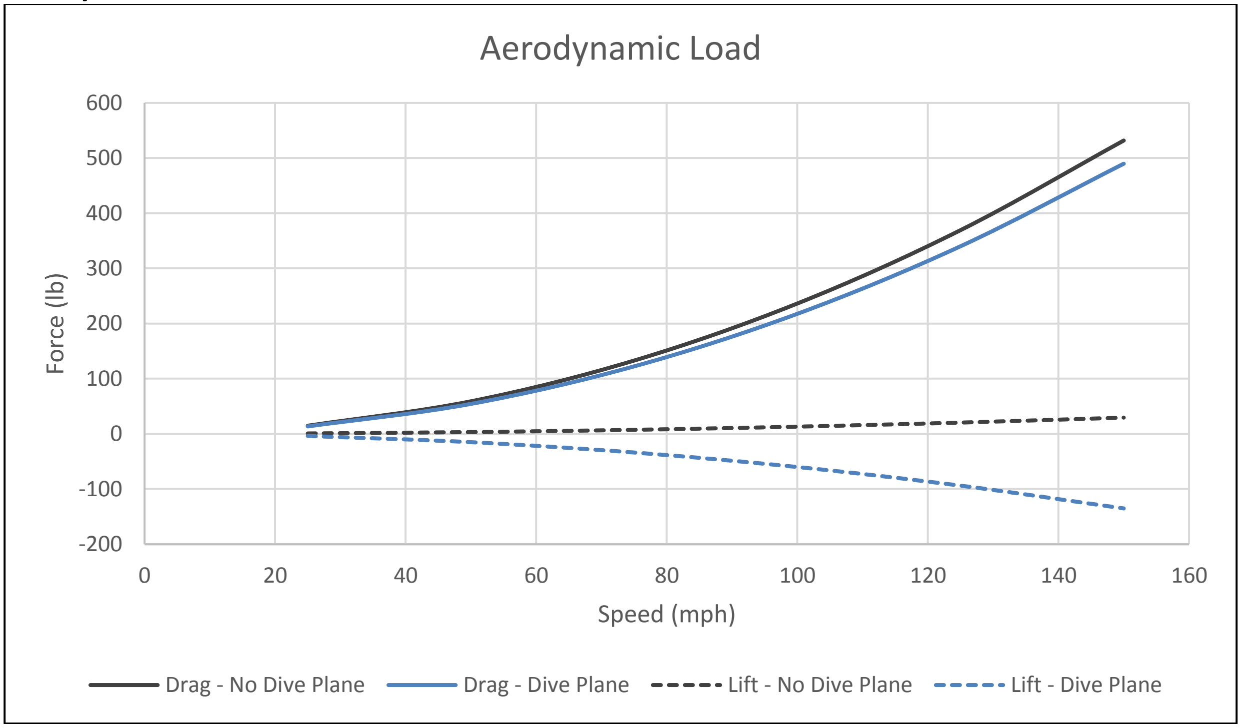
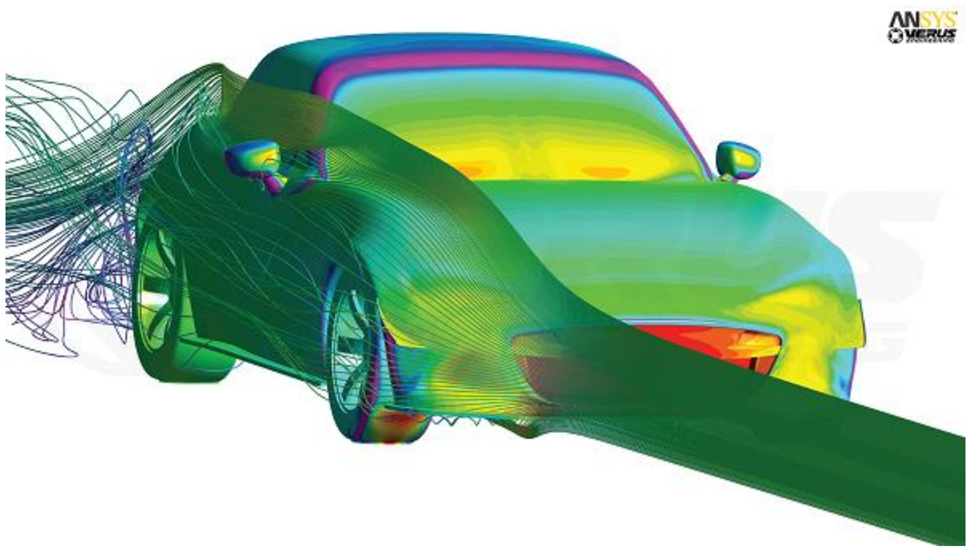


Figure 1.2.1: Aerodynamic Forces at Speed

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.

1.3. Summary Plots

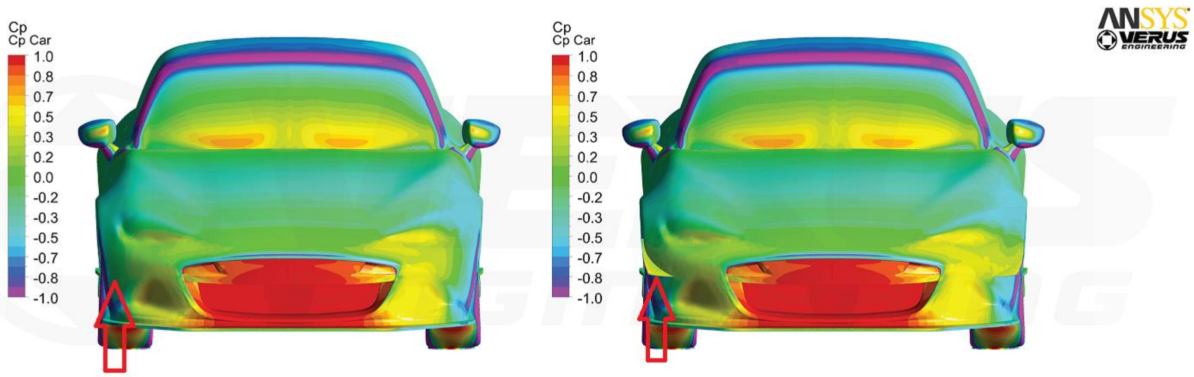


2. <u>Performance Explained</u>

2.1. Pressure Plots

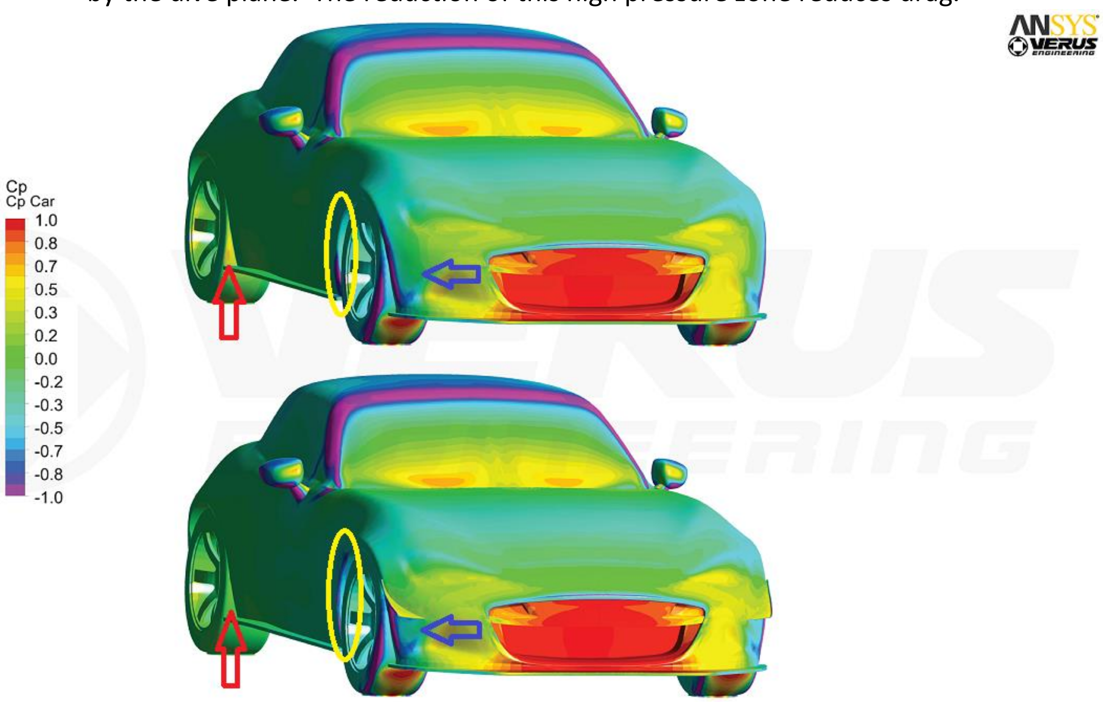
2.1.1. Front View – The red arrow below shows high pressure on the top dive plane surface. This high pressure region creates downforce.

Dive Plane Analysis on the ND Mazda Miata



2.1.2. Three quarter view:

- 2.1.2.1. The blue arrow is pointing out a location of lower pressure under the dive plane than without the dive plane. This reduces front end lift.
- 2.1.2.2. The yellow circle shows an area of low pressure behind the front wheels with the dive planes which was not there without dive planes. This signifies increased wheel well evacuation, reducing both lift and drag.
- 2.1.2.3. The red arrow is a location pointing out a high pressure region which is reduced significantly with the dive planes installed. This is from the strong vortex produced by the dive plane. The reduction of this high pressure zone reduces drag.

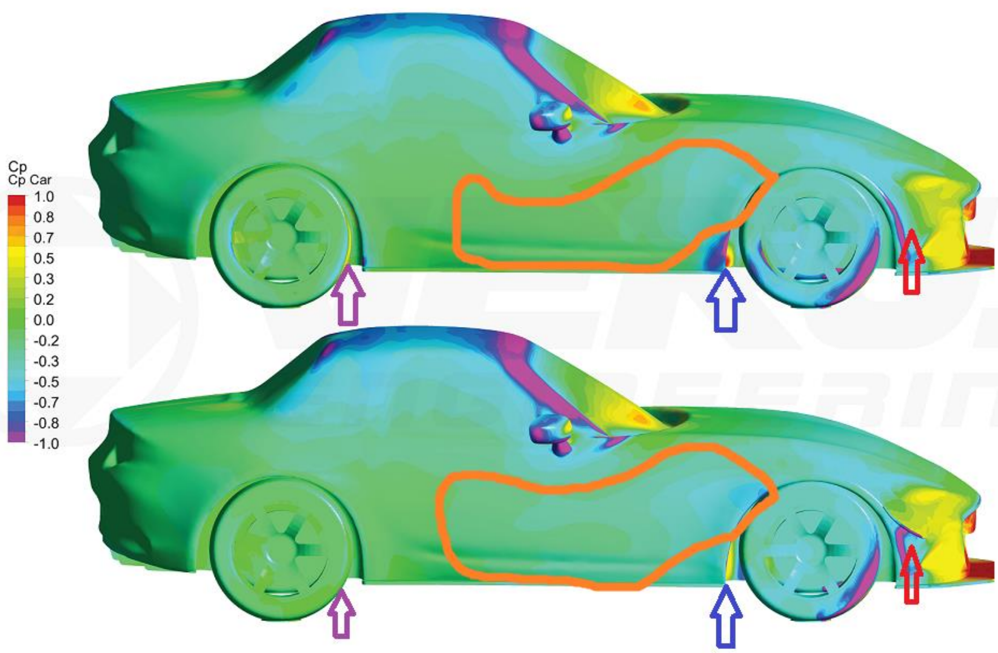


2.1.3. Side View:

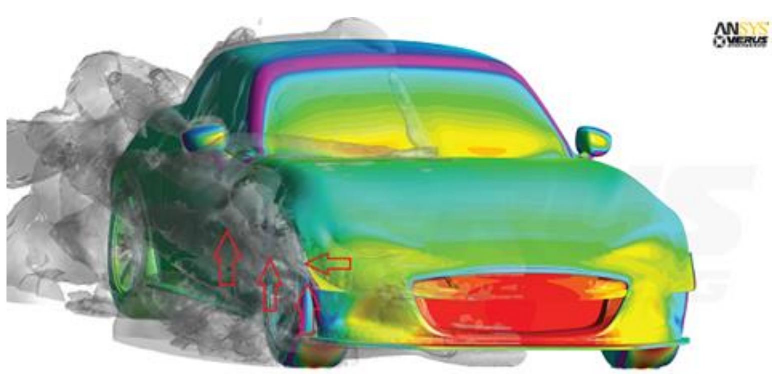
- 2.1.3.1. The purple arrow points out an area of high pressure on the tire which is reduced with the dive planes, this reduces drag.
- 2.1.3.2. The orange blob signifies a large area on the body which has reduced pressure. This reduces vehicle lift and drag as well.

2.1.3.3. The blue arrow is showing an area of high pressure in the wheel well which is reduced with dive planes. This signifies wheel well evacuation and benefits the vehicle by reducing lift in the wheel wells as well as drag reduction.

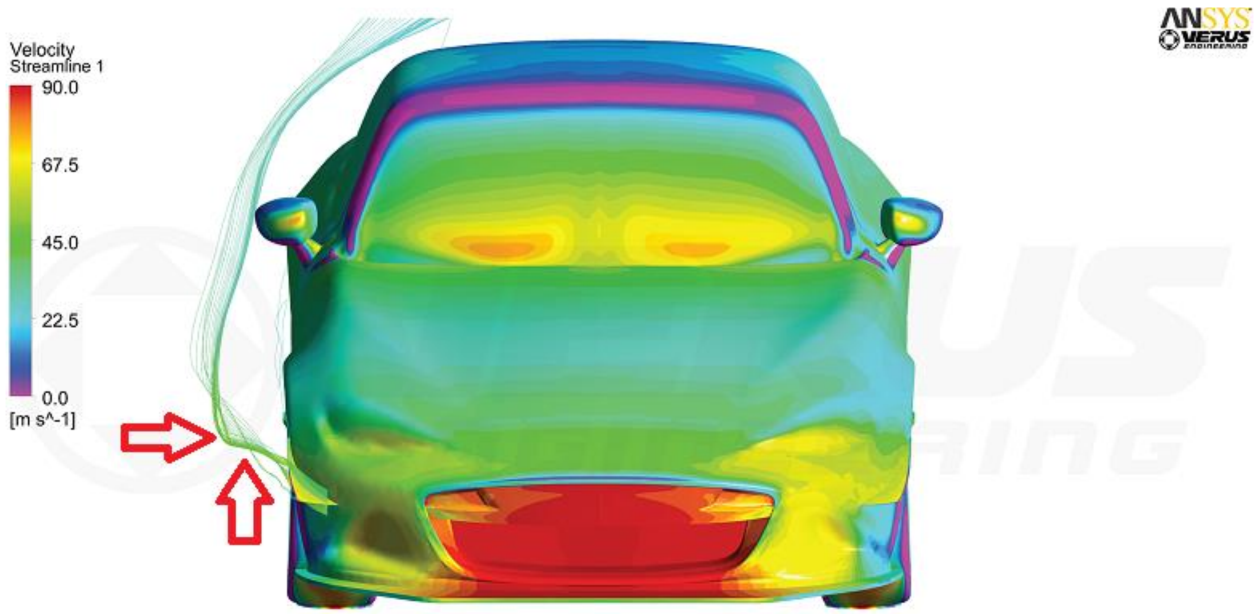
2.1.3.4. The red arrow is locates an area of low pressure under the dive plane which aids downforce production. Directly above this is a larger area of high pressure, which again aids downforce.



2.2. Vortices Plot: Q-criterion is a plot which shows vorticity, which is shown below. The red arrows below show the strong vortices coming off the dive plane which help seal the sides of the vehicle, reducing drag and lift. Q-criterion is a great way to highlight these beneficial vortices, both large and small, which is otherwise impossible or difficult to capture with other plot techniques.



2.3. Streamline Plot: Streamlines allow plotting of the flow field around the vehicle. Sometimes these streamlines can pick up strong vortices as shown below.



3. Analysis Information

3.1. Assumptions

3.1.1.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 ND Miata and how the Verus Engineering diffuser, splitter and dive planes improve the aerodynamic efficiency. The simplifications made were as follows:

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

3.1.2.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 ND Miata, 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

3.2. MESH QUALITY AND METRICS

3.2.1. 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.

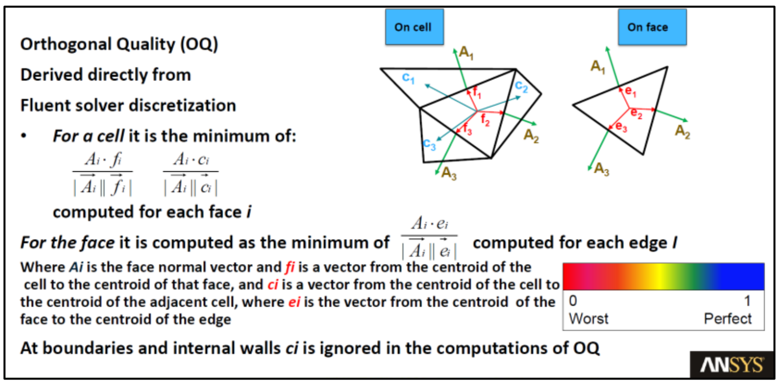


Figure 3.1.1: Orthogonal Quality Metrics

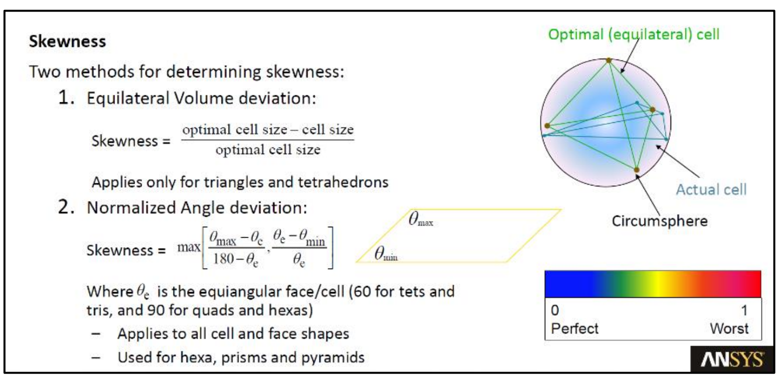


Figure 3.1.2: Skewness Quality Metrics

3.2.2. 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.

Dive Plane Analysis on the ND Mazda Miata

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

3.2.3.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.

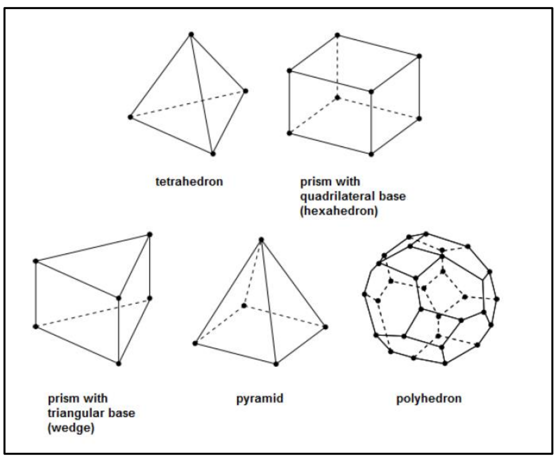


Figure 3.3.1: Volume Cell Shapes

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.

3.2.4.Mesh Quality

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

Skewness was not above 0.77. 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 16 million.

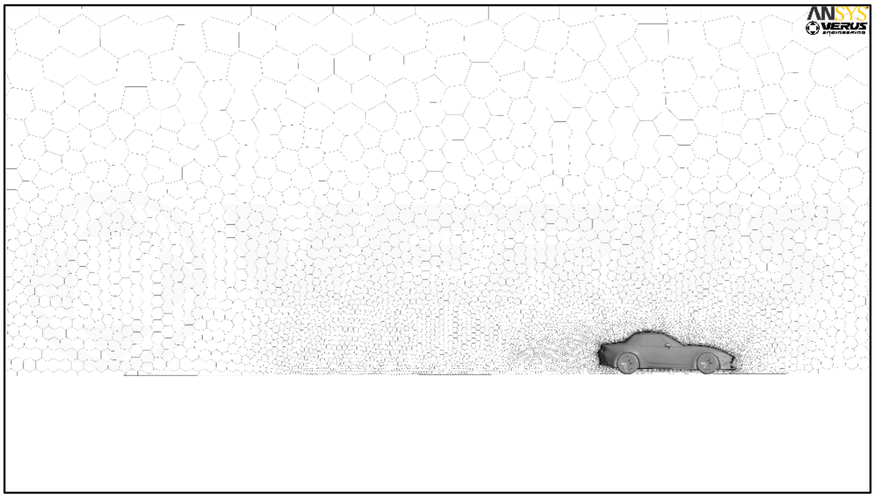


Figure 3.4.1: Mesh Domain

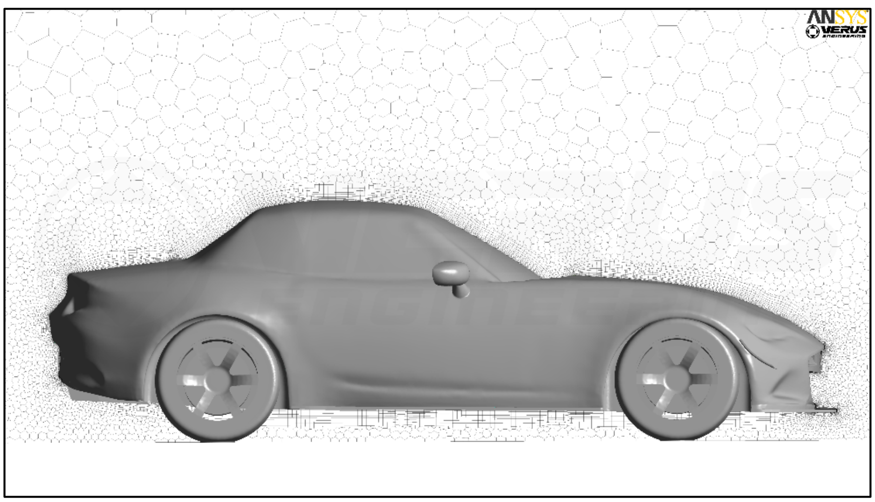


Figure 3.4.2: Mesh at Symmetry Plane

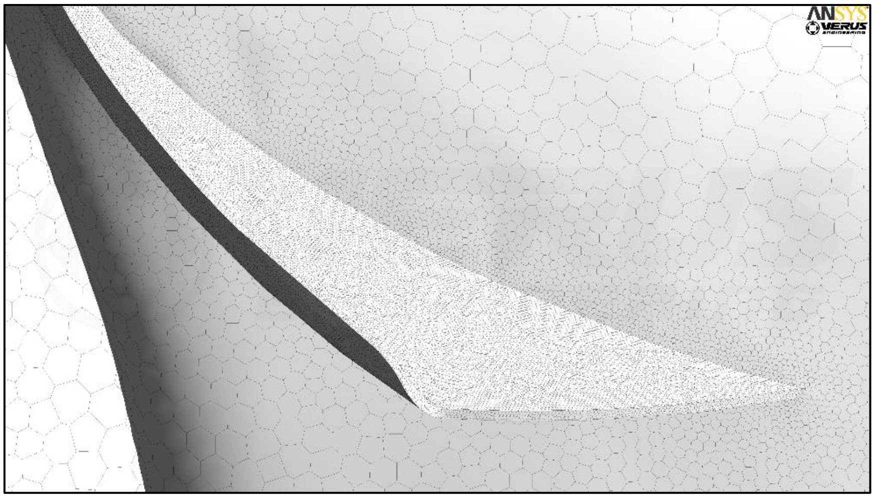


Figure 3.4.3: Surface Mesh

3.3. ANALYSIS SETUP

3.3.1. 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.

3.3.2.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	140 rad/s

Figure 4.2.1: Boundary Conditions

3.3.3. **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]

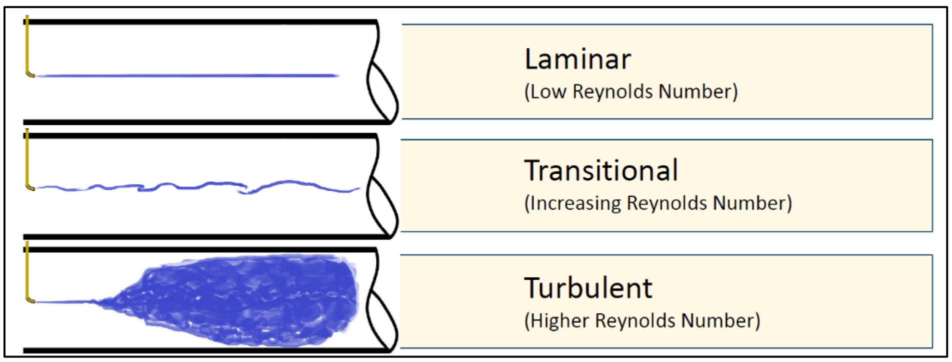


Figure 4.3.1: Flow Classifications

3.4. ERRORS IN SIMULATION

3.4.1. 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.

3.4.2. 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]

4. CONCLUSION

The goal of the dive planes were to work as a package for the ND Miata with our rear diffuser and street splitter, along with side splitters. The package is designed for the enthusiast who frequent track days and want an effective aerodynamic package that balances the car at speed. Components were designed in a way to benefit one another and keep the vehicle neutral.

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).