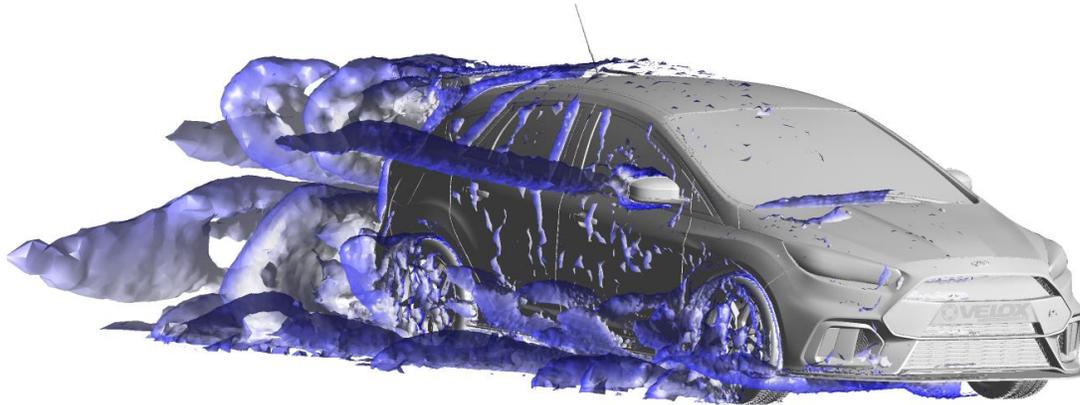


VERUS ENGINEERING

Focus RS

Performance of Verus Engineering Ventus 1 Package

ANSYS
R17.0
VELOX



Author: Paul Lucas

Release Date: 2017/03/22

Approvals: P. Lucas, E. Hazen

Document Revisions

Rev	Date	Author	Description
1	2017/03/22	P. Lucas	Issued for Release
2	2017/08/07	P. Lucas	Company Name Change From Velox to Verus

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

Focus RS – Performance of Verus Engineering Ventus 1 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	6
3.	<u>MESH QUALITY AND METRICS</u>	6
	3.1. Metrics	6
	3.2. Quality	7
	3.3. Types of Mesh	8
	3.4. Mesh Quality	8
4.	<u>ANALYSIS SETUP</u>	9
	4.1. General	9
	4.2. Boundary Conditions	9
	4.3. Turbulence Modeling	9
5.	<u>ERRORS IN SIMULATION</u>	10
	5.1. Physical Model	10
	5.2. Discretization Error	10
6.	<u>CONCLUSION</u>	10
7.	<u>REFERENCES</u>	11

Focus RS – Performance of Verus Engineering Ventus 1 Package

1. Executive Summary

- Increased downforce
- Increased drag slightly
- Improved overall aerodynamic efficiency
- Ventus 1 Package is rear diffuser, front splitter, and side splitter

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 (Fr/Rr)
+0.017	-0.115	43/57

Figure 1.1.1: Coefficient Deltas

1.2. Aerodynamic Forces

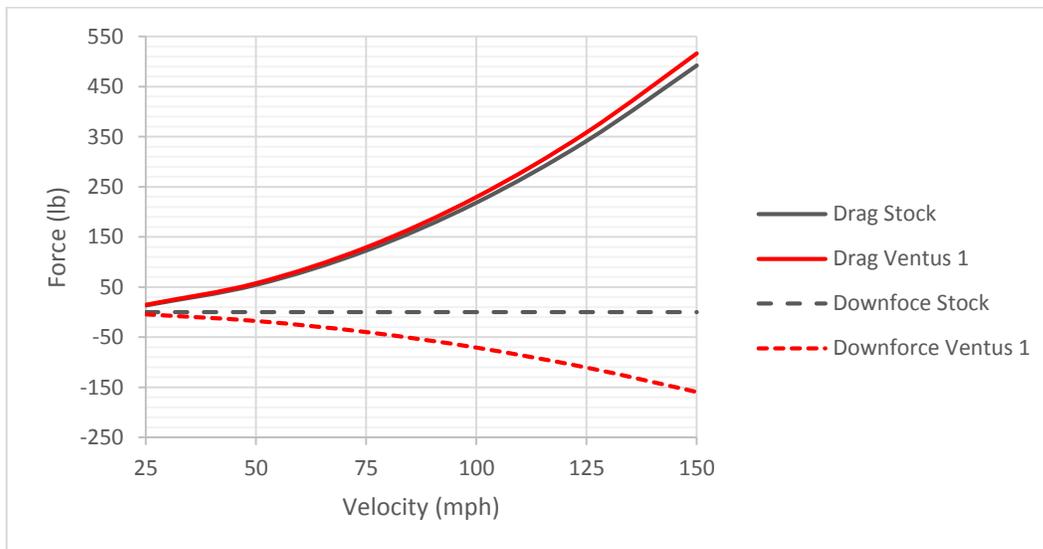


Figure 1.2.1: Aerodynamic Forces at Speed

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, however the Focus RS is not in this category. Notice Figure 5.2.1, the factory Focus RS creates no lift or downforce from factory. [1] With the Ventus 1 package, this downforce is increased quite substantially.

Focus RS – Performance of Verus Engineering Ventus 1 Package

1.3. Summary Plots

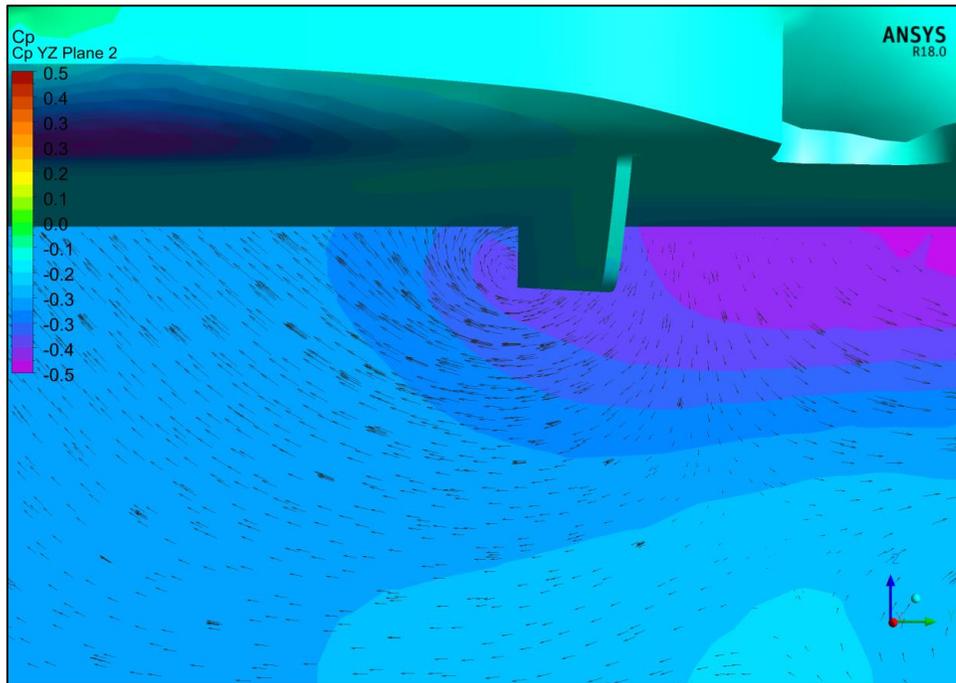


Figure 1.3.1: Strake Vortices

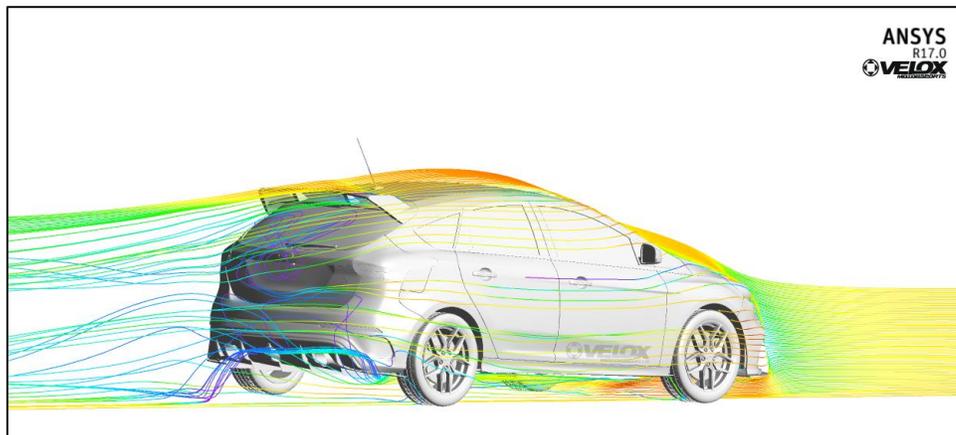


Figure 1.3.2: Streamlines from Rear Diffuser

Focus RS – Performance of Verus Engineering Ventus 1 Package

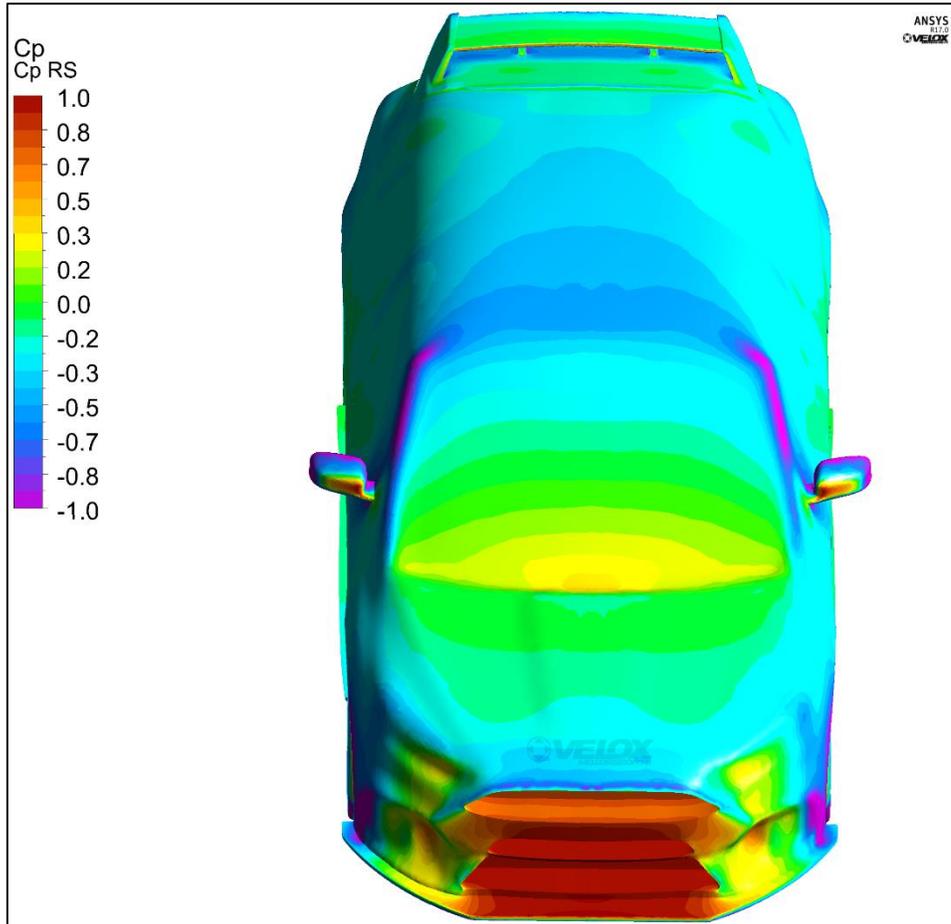


Figure 1.3.3: Surface Cp on the RS



Figure 1.3.4: Q-Criterion (vorticity)

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

Focus RS – Performance of Verus Engineering Ventus 1 Package

Notice the vortex coming off the strake in Figure 1.3.1. Vortices in these locations have a positive impact on diffuser performance. We use vortices to optimize many aspects of our components. Another way to help optimize components including a rear diffuser is streamlines as shown in Figure 1.3.2. Cp plots show pressure zones. Figure 1.3.3 shows pressure on the surface of the RS. Notice the pressure on the top side of the splitter. The Q-Criterion show vortices coming off the RS as shown in Figure 1.3.4. Q-Criterion helps locate vortices and allows us to direct vortices to locations where they can be utilized efficiently and positively.

2. ASSUMPTIONS

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. [2] 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 Focus RS and how the Verus Engineering Ventus 1 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 Ford Focus RS, 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.

Focus RS – Performance of Verus Engineering Ventus 1 Package

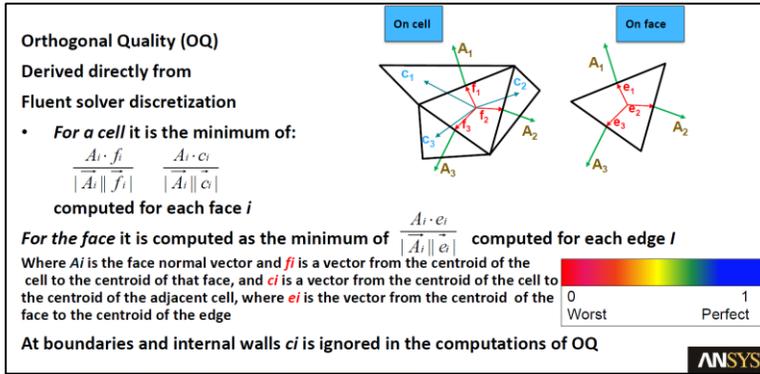


Figure 3.1.1: Orthogonal Quality Metrics

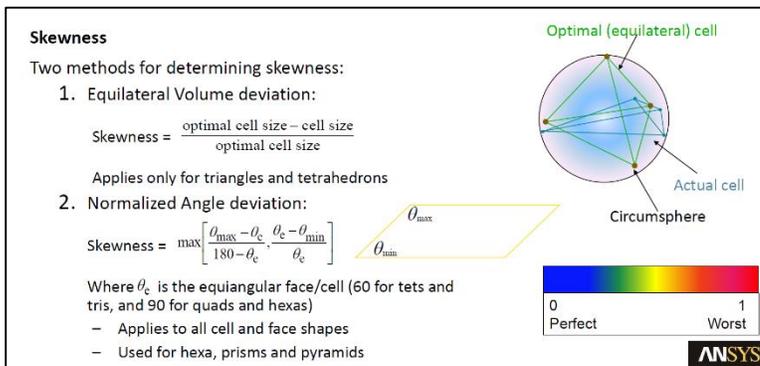


Figure 3.1.2: Skewness Quality Metrics

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.

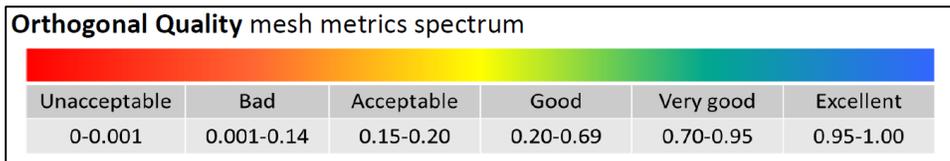


Figure 3.2.1: Orthogonal Quality Metrics

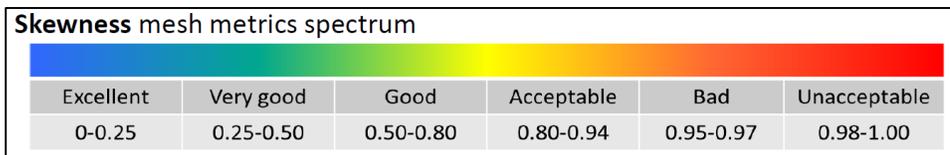


Figure 3.2.2: Skewness Quality Metrics

Focus RS – Performance of Verus Engineering Ventus 1 Package

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.

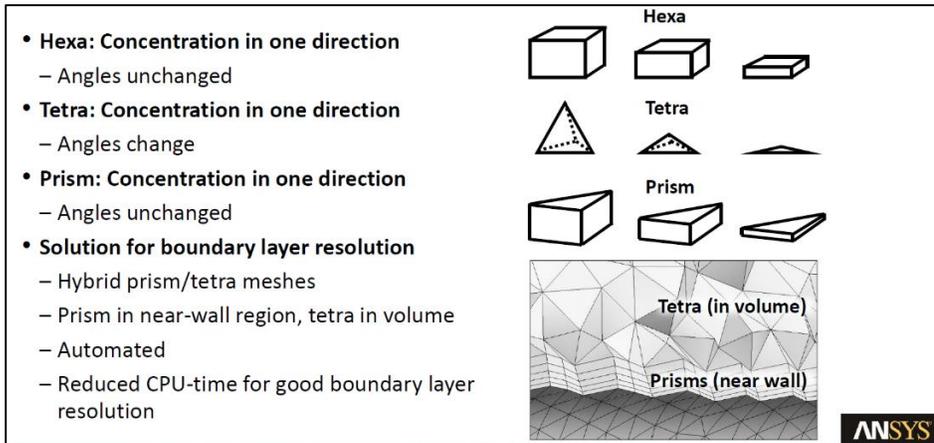


Figure 3.3.1: Hexahedral versus Tetrahedral

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

3.4. Mesh Quality

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

Orthogonal quality did not drop below 0.19. This is in the acceptable zone for the minimum orthogonal quality. The majority of the mesh is in the very good to excellent range.

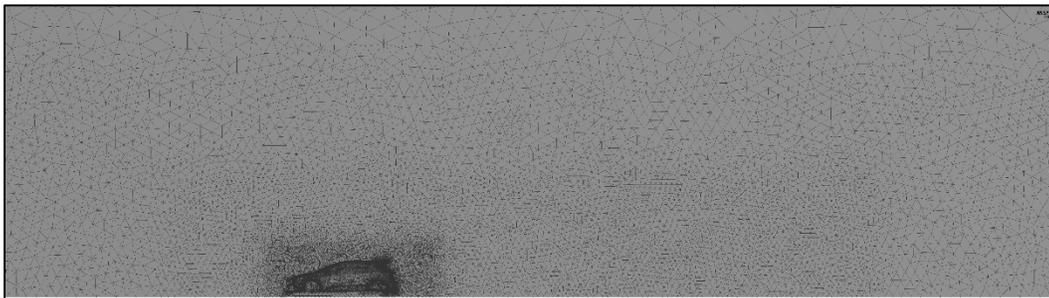


Figure 3.4.2: Mesh Domain

Focus RS – Performance of Verus Engineering Ventus 1 Package

 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	Wall Condition	Shear = 0
Focus RS	Wall Condition	No Slip Shear
Wheels	Wall Condition	141 rad/s

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]

Focus RS – Performance of Verus Engineering Ventus 1 Package

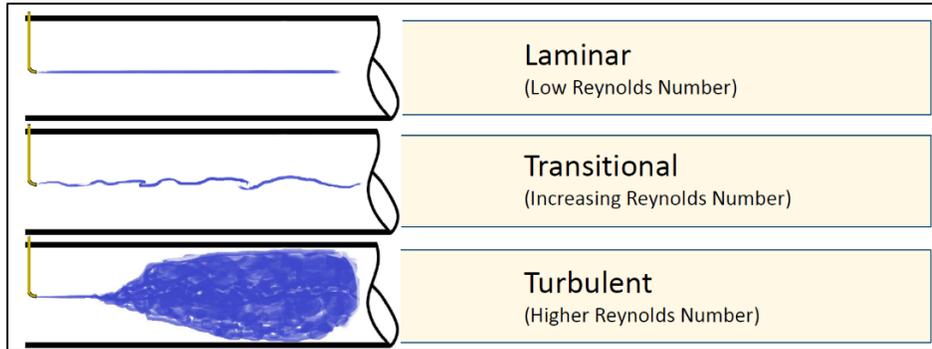


Figure 4.3.1: Flow Classifications

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 1 Package for the Ford Focus RS is to improve its aerodynamic performance over stock. The rear diffuser and front splitter were designed together to improve downforce and equal the aerodynamic balance of the factory RS. The side splitters help improve downforce and help decrease spillage of air out from under the car which improves efficiency. All the parts in the Ventus 1 Package works well as a system to improve the aerodynamics of the Ford Focus RS.

Focus RS – Performance of Verus Engineering Ventus 1 Package

7. REFERENCES

- [1] All-New Ford Focus RS Makes U.S. Debut in New York.
<https://media.ford.com/content/fordmedia/fna/us/en/news/2015/03/25/all-new-ford-focus-rs-makes-us-debut-in-new-york.html>
(Accessed 2017-03-20).
- [2] Zikanov, Oleg. *Essential Computational Fluid Dynamics*. 1st Edition. Joh Wiley & Sons, Inc., 2010.
- [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).