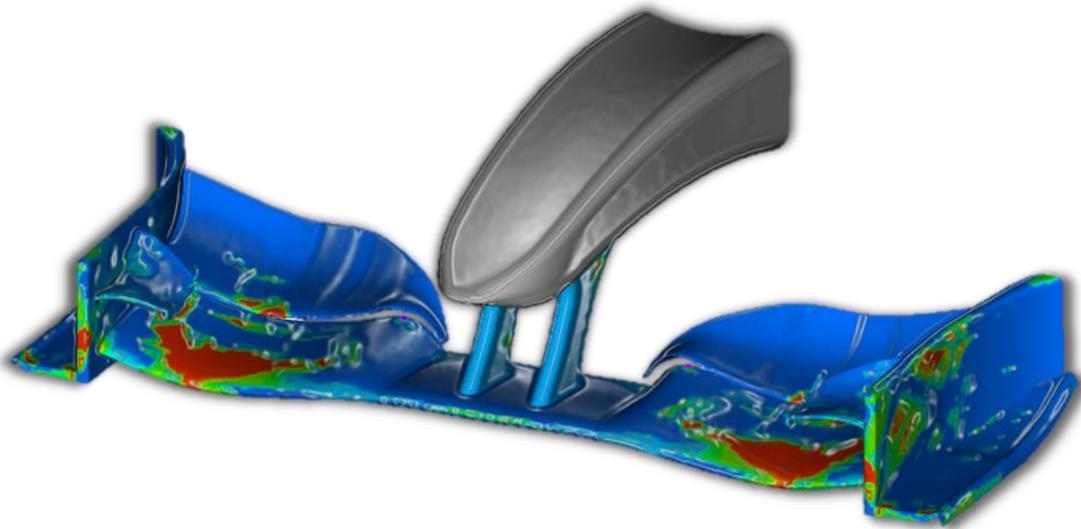# WEIGHT OPTIMIZATION OF A F1 COMPOSITE FRONT WING

[1] Ioannis Oxyzoglou, [2] Ioannis Nerantzis
[1] University of Thessaly, *Greece*, [2] BETA CAE Systems S.A., *Greece*

ABSTRACT

The aim of this project is to give the user a general idea on how a weight optimization for a geometry made out of Composite materials can be done using ANSA Laminates tools, META *post*-Processor, NASTRAN-SOL200 and modeFRONTIER optimizer. As an appropriate example, a 2011 Formula 1 front wing is selected as the model to be optimized. The main task of the optimization is to minimize the wings' total weight, while keeping it at the same time stiff enough in order to resist on the pressure distribution that is created when the car reaches the final speed of 320km/h and make it also compatible with FIA 2011 Regulations. Beginning with a CFD simulation the pressure distribution is calculated and then mapped on the FE model where it is situated as a loadcase for the FE Analysis, while for FIAs Regulation the displacement of the wing is tested under a force of 100kg. For the optimization of the wings' composite structure the layer thickness and layer orientation are used as design variables, creating each time a new laminate model while all layers are being draped after any change on the model. Finally, a comparison between the optimized model and a simple laminate model is done for the weight and the cost of each wing in order to highlight the benefits gained from the optimization process.

CONTENTS

1. INTRODUCTION

The final speed of a Formula 1 car during a race can reach up to 320km/h. On that speed, the aerodynamic forces that are beeing generated by the aerodynamic devices can be double the weight of the whole car. The front wing of a F1 car is the first aerodynamic device that interacts with the oncoming "clean" air and produces about the 30% of the total downforce. That means that the wing should be designed to be stiff enough to cope with these aerodynamic forces while having the minimum weight recuired. Starting with a CFD simulation at the final speed of a F1 car (*320 km/h*) the pressure distribution is calculated and then mapped on a structural FEA model. Two sub-cases are created and analyzed. The *1st sub-case* is about FIA's maximum displacement Rule, where the front wing must be able to withstand a *1000N* force acting on its endplate while not exceeding the maximum permitted Displacement of *10mm.* The *2nd Subcase* is about the minimization of the Composite stresses occurring on the wing due to the mapped pressure from the CFD model, using the Tsai-Wu criterion for the minimum Reserve Factor of all Layers.

In order to setup an initial Laminate model close to the optimized solution, the FEA model is firstly solved with an isotropic material(s*teel*) and then optimized with *NAS-TRAN-SOL200* in order to estimate an initial thickness distribution that is compitable with both constrains of *Subcase 1 & Subcase 2.* After setting up the Laminate model based on SOL200 thickness results solution an optimization of both its Layer thickness and Layer orientation is done using *EPILYSIS* and *modeFRONTIER* giving one optimal solution for the weight of the F1 wing. It is important to mention that when it comes to Composite materials the number of optimized solutions can be surprisingly high, due to the fact that the combination of different layer thickness, layers orientation and even layers' order can be infinite.
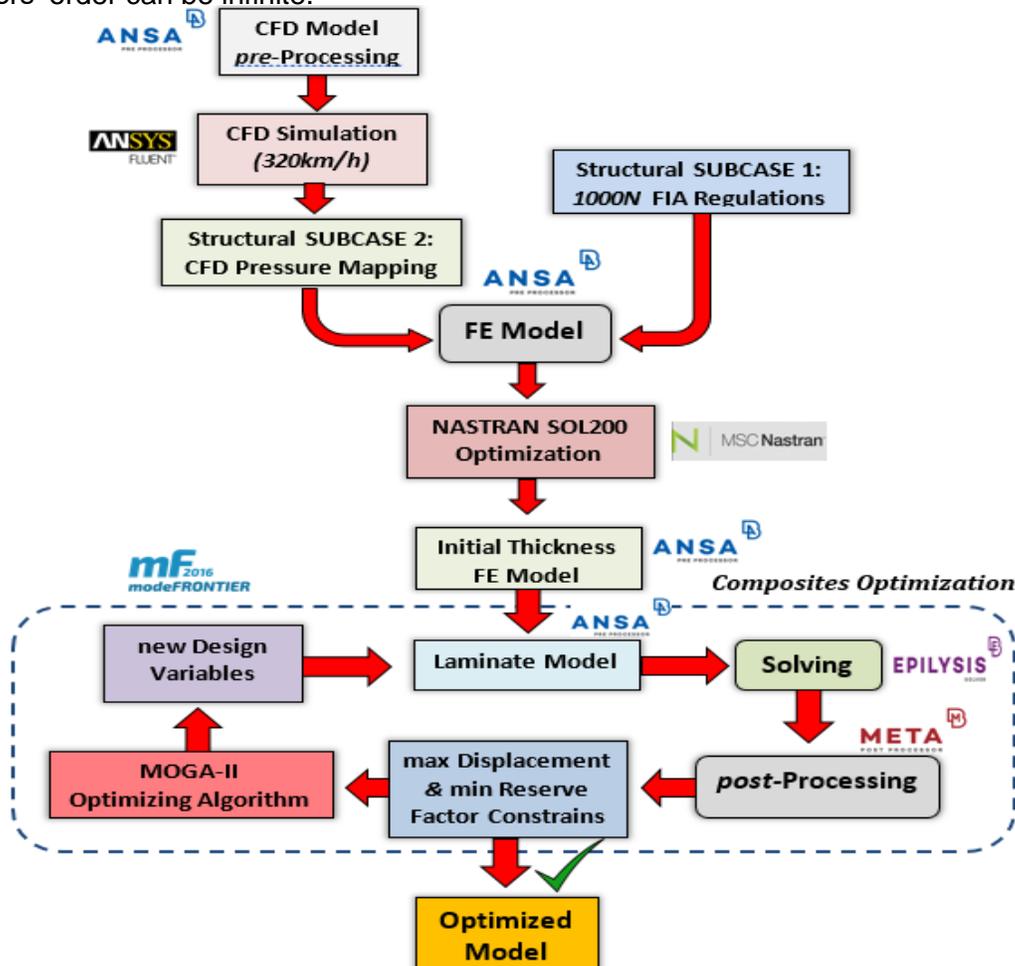
2. CFD SIMULATION

2.1. Fluid Domain & Surface Mesh

The final speed of a F1 car during a race is about 320km/h and this is also the speed where downforce reaches its maximum value. If the front wing is designed to withstand the forces acting at that speed, it will be also safe to operate at any other speed. So, in order to calculate the maximum air pressure distribution, on the wing's surface the CFD simulation is settled with an inlet speed of 320km/h (*88.88 m/s*). Due to the symmetric geometry of the wing the CFD model is settled with the symmetric half of the wing, in order to minimize the required CPU time. The fluid domain shown in **Figure 1** is extended 5 times the characteristic length of the wing to the front, 10 times to the rear and 3 times to the side in order to give enough space for the wake turbulence to be calculated and predict drag acurately. The surface mesh is created using the CFD algorithm with triangle elements of a minimum target length of 50mm, a maximum target length of 150mm, a growth rate of 1.2 and a distortion angle of 10°. Based on the Fluent Skewness criterion the shell elements should not exceed a skewness of 0.5, and as shown in **Table 1** all shell elements are bellow this limit.
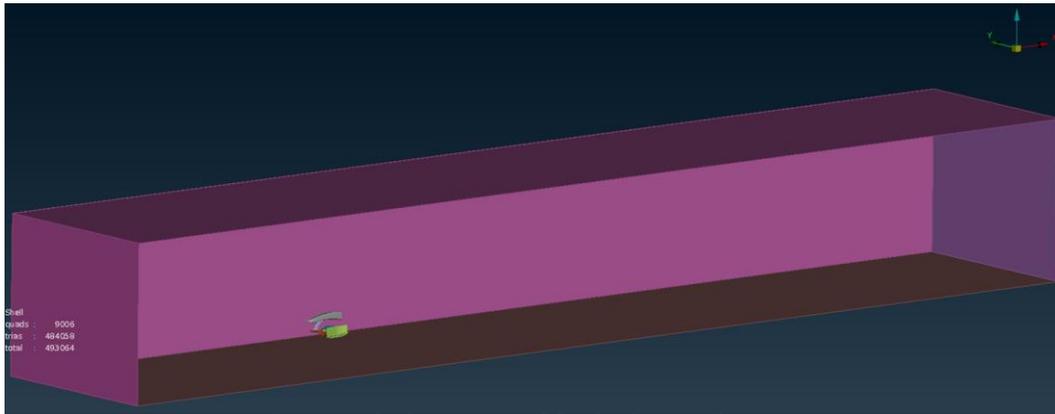

Figure 1: *Fluid Domain used for the CFD simulation*

OVERALL NUMBERS of SHELL ELEMENTs / PERCENTAGE (%)						
TYPE	**TRIA3**	**TRIA6**	**QUAD4**	**QUAD8**	**POLYGONS**	**TOTAL**
NUMBER	483948	0	0	0	0	483948
PERC %	100.000	0.000	0.000	0.000	0.000	100.000

Shell Skewness Graph

Classes	Ranges		Elements
Class 1:	< 0.00	0.00%	0
Class 2:	0.00 - 0.10	98.98%	479021
Class 3:	0.10 - 0.20	0.94%	4571
Class 4:	0.20 - 0.30	0.02%	115
Class 5:	0.30 - 0.40	0.04%	203
Class 6:	0.40 - 0.50	0.01%	38
Class 7:	0.50 - 0.60	0.00%	0
Class 8:	0.60 - 0.70	0.00%	0
Class 9:	0.70 - 0.80	0.00%	0
Class 10:	0.80 - 0.90	0.00%	0
Class 11:	0.90 - 0.92	0.00%	0
Class 12:	0.92 - 0.95	0.00%	0
Class 13:	0.95 - 0.98	0.00%	0
Class 14:	0.98 - 0.99	0.00%	0
Class 15:	0.99 - 1.00	0.00%	0
Class 16:	1.00 - 1.00	0.00%	0
Class 17:	> 1.00	0.00%	0
			Total: 483948

Table 1: *Number of shell elements based on ther Skewness value*

2.2. Size Boxes

Three size boxes are used to refine mesh for specific regions of the geometry, where stall or strong vortices are expected to occur. The element length on these size boxes is varying between 10mm - 35mm while growth rate and distortion angle is again 1.2 and 10° respectively.

Figure 2: *1ˢᵗ Size Box with max surface and volume element length 20-35mm*
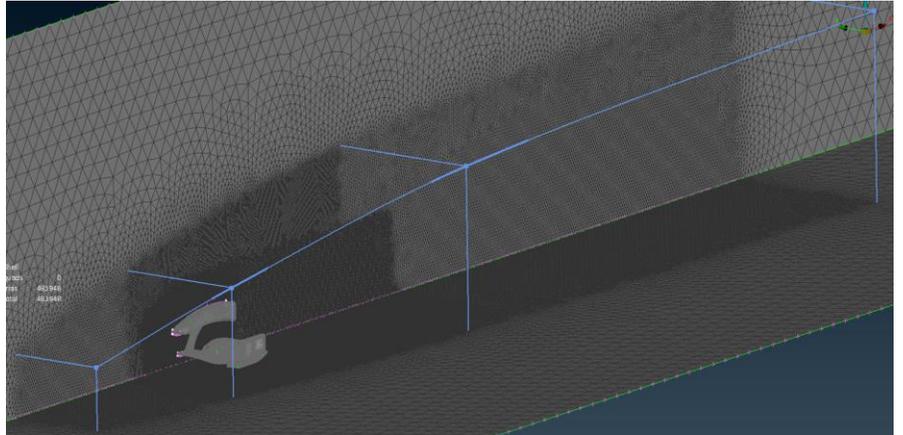

Figure 3: *1ˢᵗ Size Box with max surface and volume element length 10-20mm*
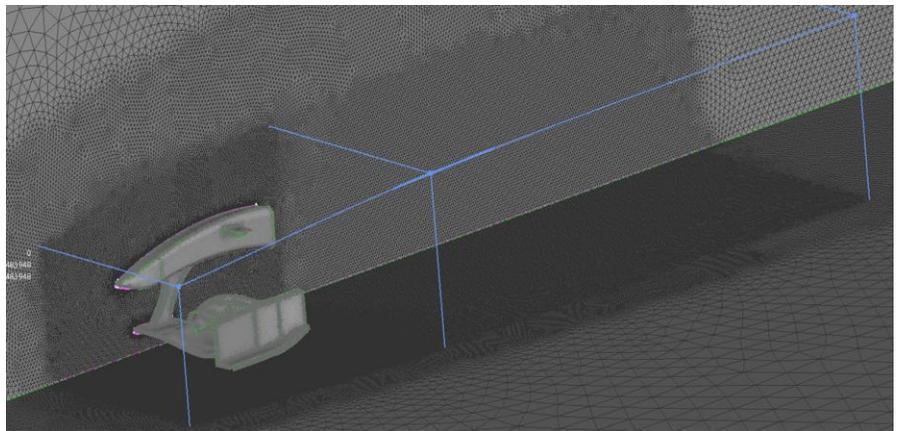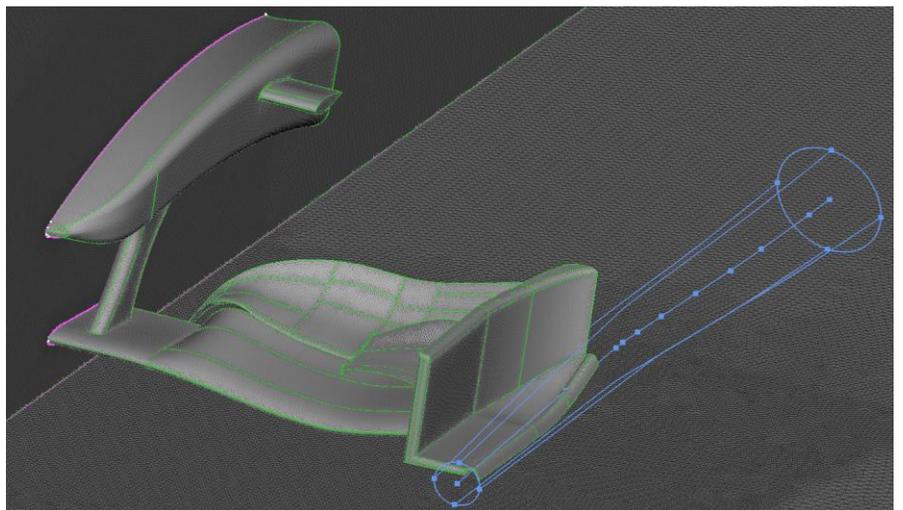

Figure 4: *3ʳᵈ Size Box with max surface and volume element length 10-15mm used to predict vortices created by the voretx generator at the bottom of the endplate*

2.3.   Layers

After the creation of the surface mesh and before the volume mesh, a series of layers has to be created in order to predict the boundary layer thickness acuurately. The estimated first layer thickness for 320km/h and a characteristic length of 0.9m, is calculated with ANSA Y+ Calculator at about *0.14mm* while Reynold's number is 5.5e+6. So, a total number of six layers created, giving the first three an absolute height of 0.15mm and the next three a growth factor of 1.2 shown in *Figure 7*.

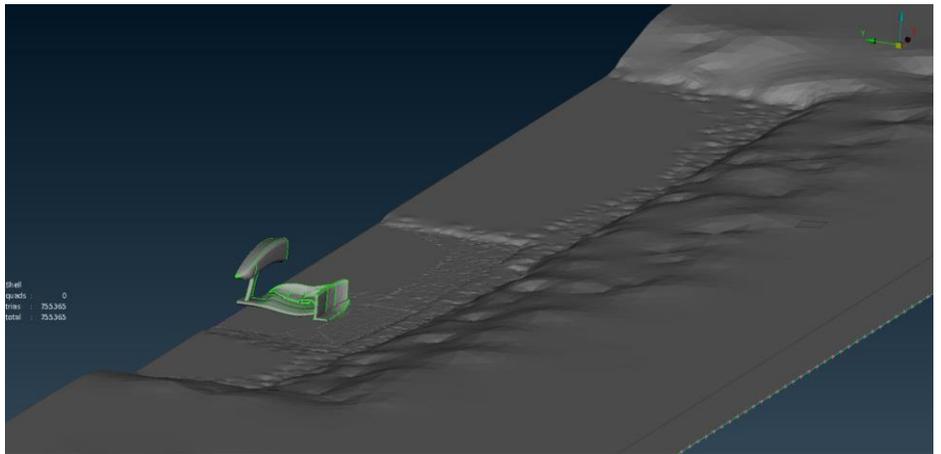**Figure 5:** *Layers generated on the interior of the fluid domain*

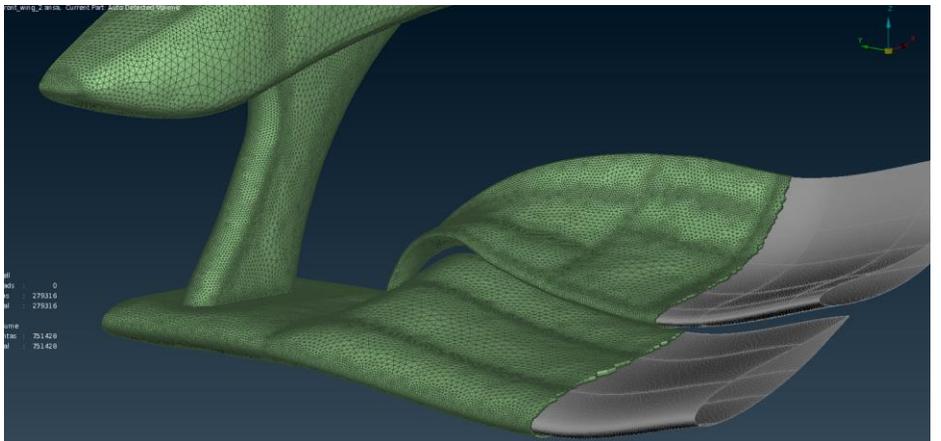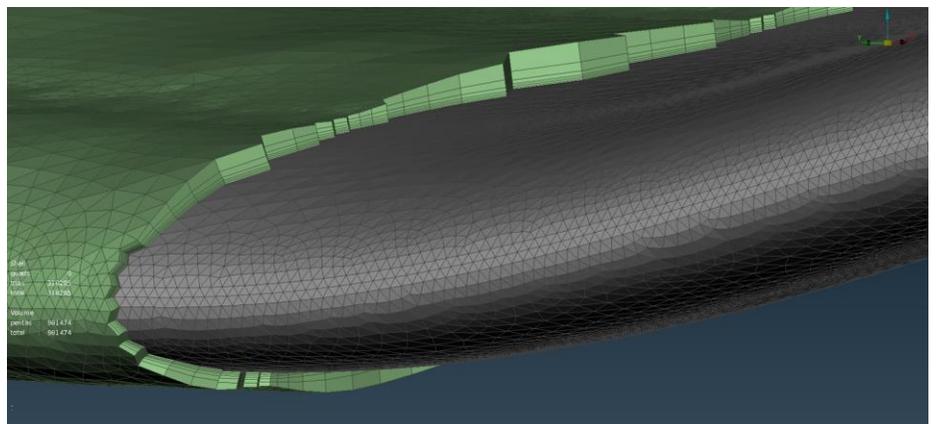**Figure 6:** *Layers covering all the wing's surfaces*

Figure 7: *Aspect layer of layers shown on detail*

2.4. Volume Mesh

The final step is the creation of the volume mesh. Using the Mesh Vol.>Tetra Rapid algorithm a volume mesh of 20,483,258 elements in total is created, with 18,222,158 tetras, 2,260,346 pentas and 754 pyramids. The Fluent skewness criterion for solid elements has an upper limit of 0.92. Based on this criterion and using ANSA mesh improvement functions (*Fix Quality, Reconstruct, Smooth*) for volume mesh, all elements are bellow 0.92 as shown in **Table 2**.

	OVERALL NUMBERS of SOLID ELEMENTs / PERCENTAGE (%)									
TYPE	**TETRA**	**TETRA2**	**PENTA**	**PENTA2**	**CHEXA**	**CHEXA2**	**PYRAMID**	**PYRAMID2**	**POLYHEDRAL**	**TOTAL**
NUMBER	18222158	0	2260346	0	0	0	754	0	0	20483258
PERC %	88.961	0.000	11.035	0.000	0.000	0.000	0.004	0.000	0.000	100.000

Figure 8: *Volume Mesh created for the CFD simulation*
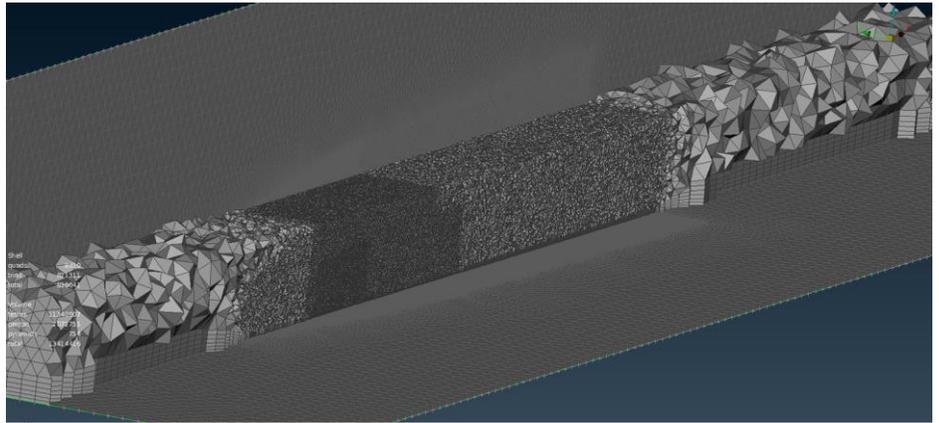
Figure 9: *Volume Mesh elements shown in detail*
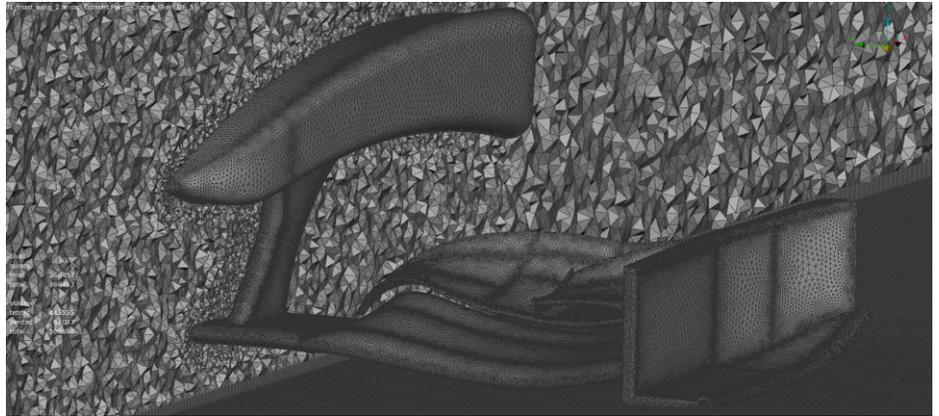
Solid Skewness (Tetras only) Graph

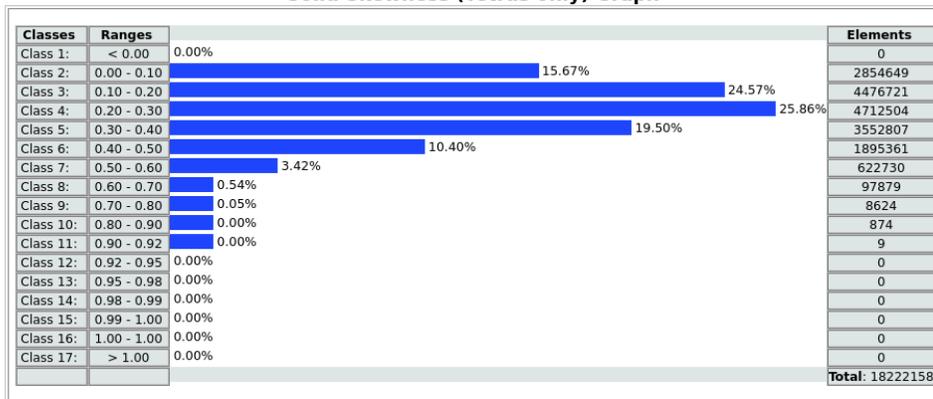| Classes	Ranges		Elements
Class 1:	< 0.00	0.00%	0
Class 2:	0.00 - 0.10	15.67%	2854649
Class 3:	0.10 - 0.20	24.57%	4476721
Class 4:	0.20 - 0.30	25.86%	4712504
Class 5:	0.30 - 0.40	19.50%	3552807
Class 6:	0.40 - 0.50	10.40%	1895361
Class 7:	0.50 - 0.60	3.42%	622730
Class 8:	0.60 - 0.70	0.54%	97879
Class 9:	0.70 - 0.80	0.05%	8624
Class 10:	0.80 - 0.90	0.00%	874
Class 11:	0.90 - 0.92	0.00%	9
Class 12:	0.92 - 0.95	0.00%	0
Class 13:	0.95 - 0.98	0.00%	0
Class 14:	0.98 - 0.99	0.00%	0
Class 15:	0.99 - 1.00	0.00%	0
Class 16:	1.00 - 1.00	0.00%	0
Class 17:	> 1.00	0.00%	0
			Total: 18222158

Table 2: *Number of volume elements based on ther Skewness value*

2.5. Solving Process

The CFD Case is finally solved by using ANSYS-*Fluent solver.* For the Boundary Conditions that are used for this CFD Simulations, an Inlet Velocity of 89m/s (*320km/h*) is defined which is also the velocity of the road which is settled as a moving wall with the same direction as the inlet velocity. The outlet of the domain is settled as a simple Pressure outlet while the rest walls of the fouid domain are defined as Symmetries. Finally, all the surfaces of the wing are defined as Stationary Walls. The turbulence model that is used is the k-ε, which is the most common model for vehicle external aerodynamics problems while for the boundary condition of the walls, the Non-Equilibrium Wall Functions are used. The case is solved by using an i7 Desktop Computer with 64GB of RAM and after a total number of 545 iterations Convergence is achieved, while the total CPU time recuired is about 18 hours.
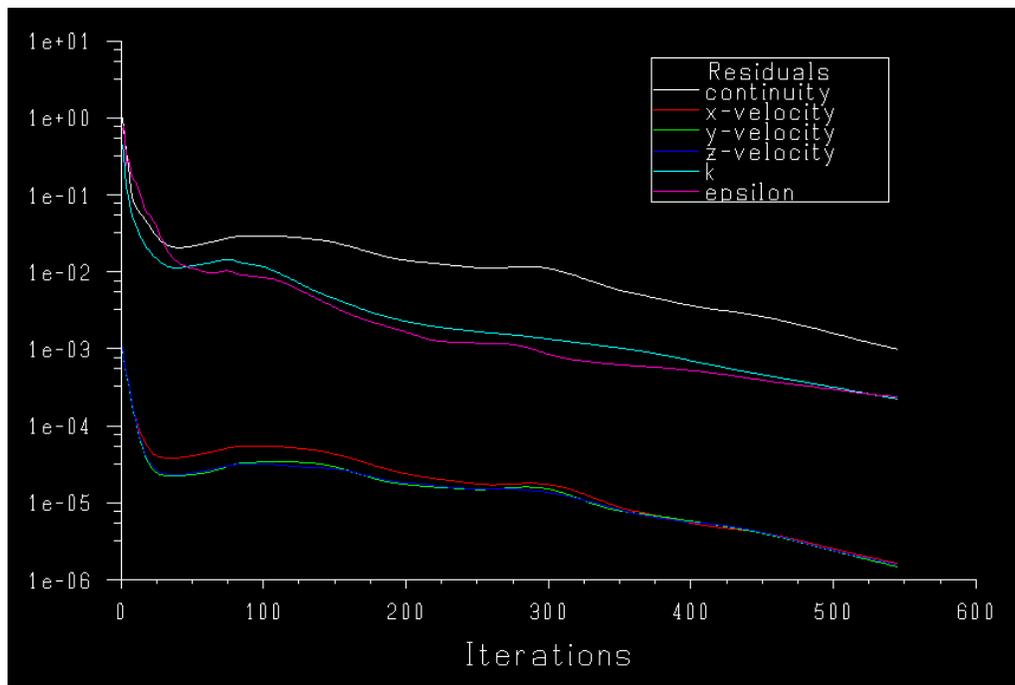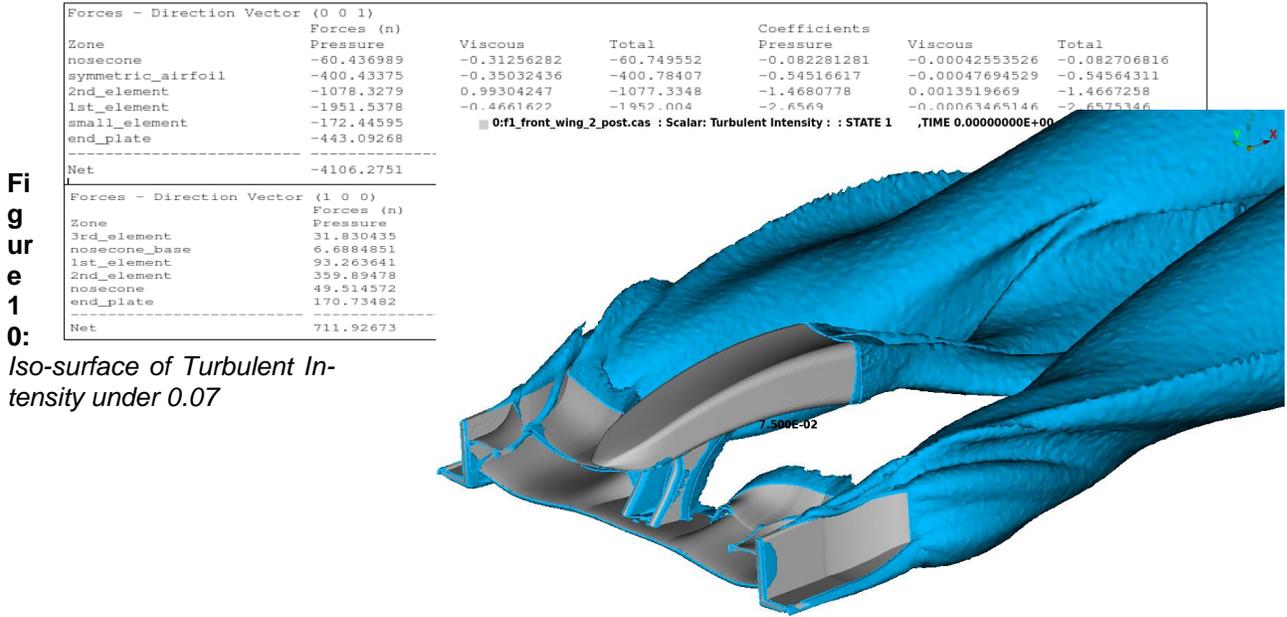

Chart 1: *After 545 Iterations Convergence is achieved*

8

2.6. CFD Results

The CFD simulation is solved with ANSYS Fluent on an i7 Desktop Computer with a RAM memory of 64GB. Total CPU time for surface & volume mesh creation is 30min while solving lasts for 14hours and after 550 iterations convergence is achieved after a total number of 550 iterations. The total downforce acting on the half wing calculated value is 4106*N* with a lift coefficient(*Cl*) of -5.59, while drag calculated value is *741N* with a drag coefficient(*Cd*) of 1.098 as shown in two tables below.

```
Forces - Direction Vector (0 0 1)
                      Forces (n)                                   Coefficients
Zone                  Pressure        Viscous         Total        Pressure        Viscous         Total
nosecone              -60.436989      -0.31256282     -60.749552   -0.082281281    -0.00042553526  -0.082706816
symmetric_airfoil     -400.43375      -0.35032436     -400.78407   -0.54516617     -0.00047694529  -0.54564311
2nd_element           -1078.3279      0.99304247      -1077.3348   -1.4680778      0.0013519669    -1.4667258
1st_element           -1951.5378      -0.4661622      -1952.004    -2.6569         -0.00063465146  -2.6575346
small_element         -172.44595
end_plate             -443.09268
                      -------------   -------------
Net                   -4106.2751

Forces - Direction Vector (1 0 0)
                      Forces (n)
Zone                  Pressure
3rd_element           31.830435
nosecone_base         6.6884851
1st_element           93.263641
2nd_element           359.89478
nosecone              49.514572
end_plate             170.73482
                      -------------
Net                   711.92673
```


Figure 10: *Iso-surface of Turbulent Intensity under 0.07*

Figure 11: *Velocity streamlines*

0:f1_front_wing_2_post.cas : Scalar: Pressure : : STATE 1   ,TIME 0.00000000E+00

Figure 12: *Pressure distribution on the upper surface of the wing*

>5024.56	
-1961.42	
-8947.4	
-15933.4	
-22919.4	
-29905.3	
-36891.3	
-43877.3	
-50863.3	
-57849.3	
<-64835.3	
No Value	

0:f1_front_wing_2_post.cas : Scalar: Pressure : : STATE 1   ,TIME 0.00000000E+00
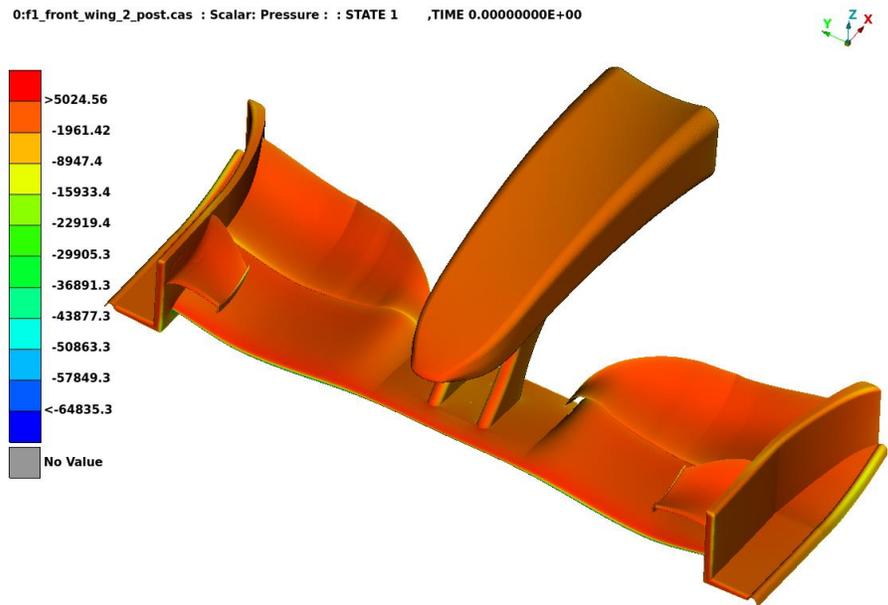
Figure 13: *Pressure distribution on the downside surface of the wing*

>5024.56	
-1961.42	
-8947.4	
-15933.4	
-22919.4	
-29905.3	
-36891.3	
-43877.3	
-50863.3	
-57849.3	
<-64835.3	
No Value	
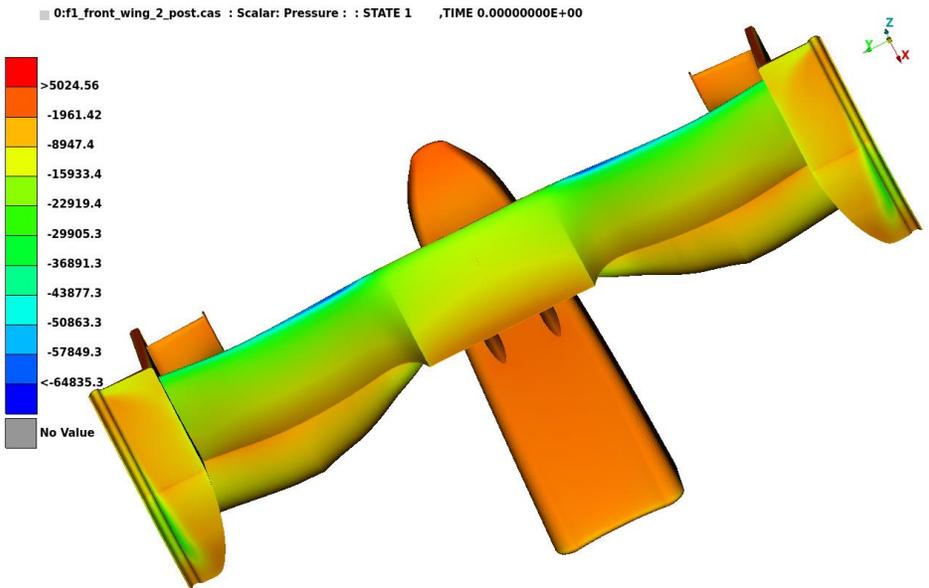
3. FEA MODEL SETUP

3.1. Surface Mesh & Problem Constrains

As the CFD simulation is completed and pressure distribution on the wing calculated, the FEA model is ready to be situated. Again, half of the wing is used as model due to the symmetric geometry, in order to minimize optimization's process time. Both isotropic and composite models are going to be solved with NASTRAN_SOL200 and EPILYSIS so the FEA model is based on NASTRAN solver. Shell mesh is created using ANSA Free meshing algorithm with quad element type, occurring in a total number of 84,743 elements, out of which 83,522 are quads and 1,221 trias. Aspect Ratio (*3*), Skewness(*45*), Warping(*15*), min Angle for quads(*45*), max Angle for quads(*135*), min Angle for trias(*30*), max Angle for trias(*120*) and triangles percent(*20%*) used as mesh Quality criteria for the FEA model and as shown *Table 3* and *Table 4* below all elements are beneath the limits.

Figure 13: *Total Pressure distribution on the downside surface of the wing*

Shell Aspect Graph

Total: 83522	Class 1: 0	Class 2: 83125	Class 3: 397	Class 4: 0	Class 5: 0	Class 6: 0	Class 7: 0
	0.00%	99.52%	0.48%	0.00%	0.00%	0.00%	0.00%
	< 0.00	0.00 - 2.00	2.00 - 3.00	3.00 - 4.00	4.00 - 5.00	5.00 - 10.00	> 10.00
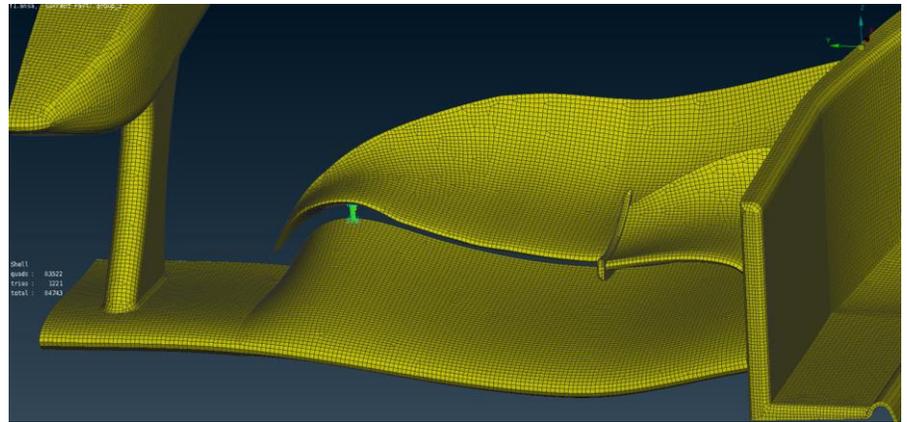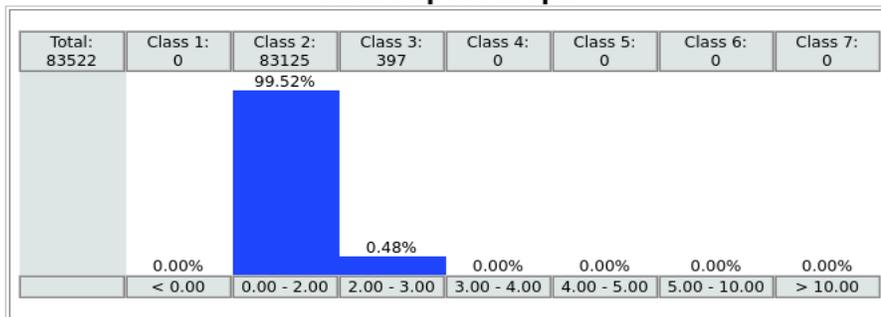
Table 3: *Number of shell elements complying with aspect ratio criterion*

Shell Skewness Graph

Total: 84743	Class 1: 0	Class 2: 84022	Class 3: 685	Class 4: 36	Class 5: 0	Class 6: 0	Class 7: 0
	0.00%	99.15%	0.81%	0.04%	0.00%	0.00%	0.00%
	< 0.00	0.00 - 18.00	18.00 - 36.00	36.00 - 45.00	45.00 - 72.00	72.00 - 90.00	> 90.00
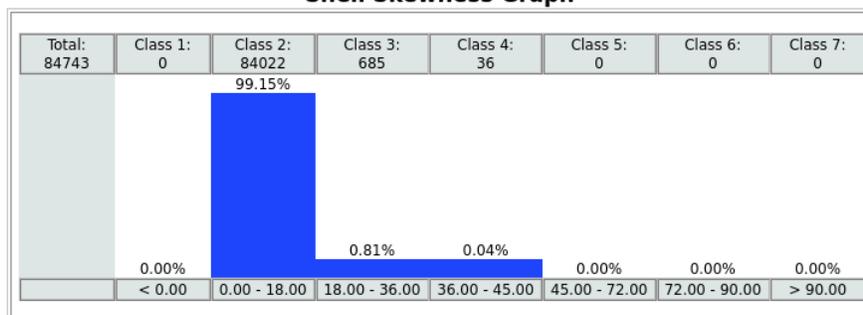
Table 4: *Number of shell elements complying with skewness criterion*

Shell Warping Graph

Total: 83522	Class 1: 0	Class 2: 82154	Class 3: 843	Class 4: 233	Class 5: 95	Class 6: 197	Class 7: 0
	0.00%	98.36%	1.01%	0.28%	0.11%	0.24%	0.00%
	< 0.00	0.00 - 2.00	2.00 - 4.00	4.00 - 6.00	6.00 - 8.00	8.00 - 10.00	> 10.00
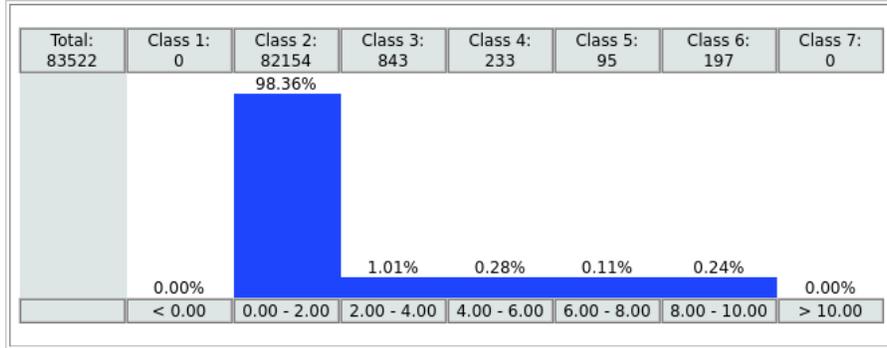
Table 5: *Number of shell elements complying with warping criterion*

Two sets of SPC1 constrains are created in order to make the symmetric problem feasible. One SPC1 set for the rear end of the model with 0 degrees of freedom (*1,2,3,4,5,6*) and the other one is at the symmetric nodes at the cutting edge with 3 degrees of freedom (*2,4,6*) as shown in next *Figure 14* and *Figure 15*.

Figure 14: *Rear side SPC1 set*
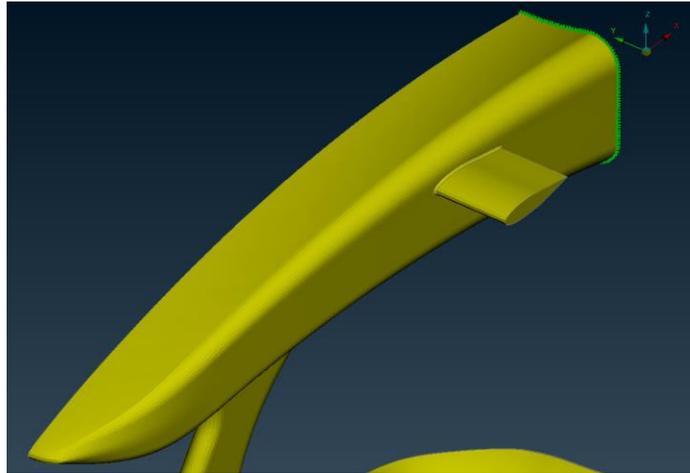
Figure 15: *Symmetric side SPC1 set*
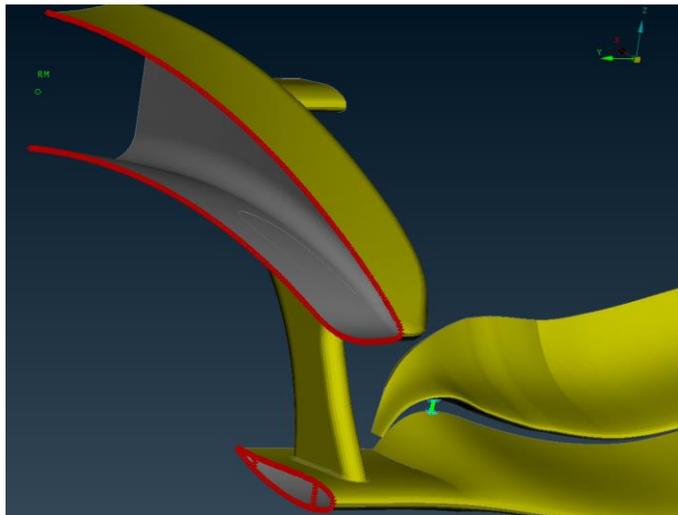
Most F1 teams use small connecting components between the airfoils in order to make the whole construction more stable and reduce the composite stresses. For this project a small cylindrical bar used to connect the two airfoils (**Figure 16**) in a way that stresses on both airfoils are minimizing. In order to simulate this type of connection, two sets of 9 nodes for each airfoil are defined as RBE2 rigid body of elements while the master nodes connected with a cylindrical CBAR of *4mm* diameter. On the interior of the airfoils, a complex of 5 ribs and 3 spars (**Figure 17**) made of steel are also used in order to support the whole structure and improve durability.

Figure 16: *CBAR used for the connection of the two airfoils*
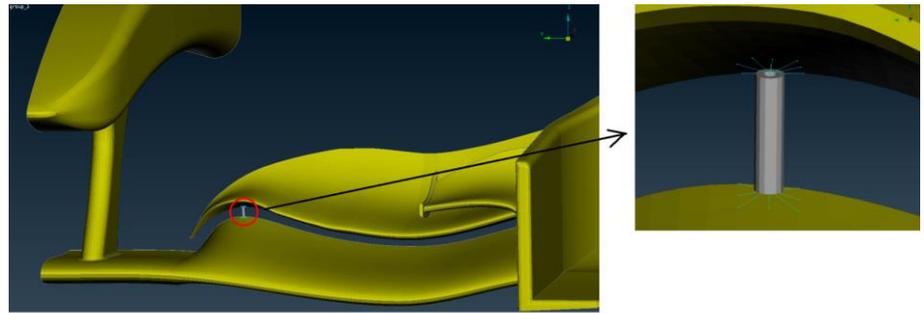
Figure 17: *Spars & Ribs used as support for the main structure*
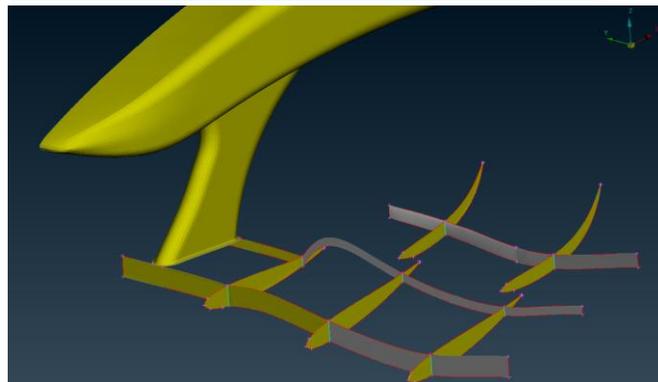
3.2. Loadcases

Next is the definition of the loads and the creation of each separate Subcase. As already mentioned the *Subcase_1* is about FIAs 2011 Rules, where it is stated that when a total force of *1000N* is acting on the endplate of the wing the total displacement of the wing's lowest point should not exceed *10mm.* To simulate this a force of 1000N is situated on the upper surface of the endplate and by using *DistributeLoad.py* script this force is distributed equivalent on all elements of the surface as shown in **Figure 18**.

Figure 18: *Equal distributed 1000N force for Subcase_1 (FIA Rules)*
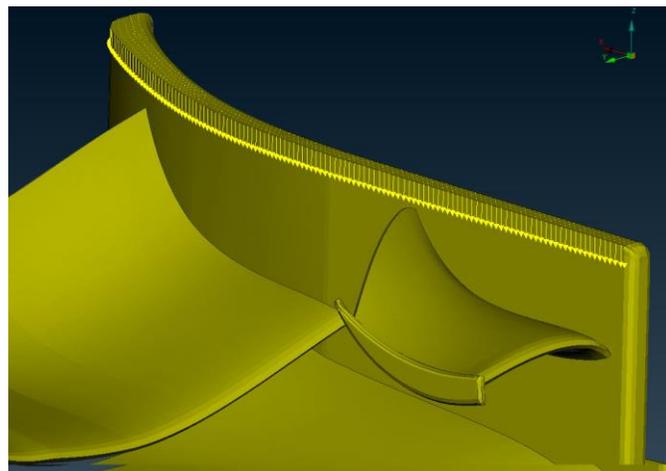
The aim of Subcase_2 is to test the durability of the wing on the acting air pressure that is calculated at the final speed of *320km/h.* In order to transfer the precise pressure distribution from the CFD model into the FEA model, the pressure results had to be mapped and translated to NASTRAN PLOAD2 pressure. After reading the pressure results in mETA, an output file in .dat (*or .txt*) form is created including the nodal pressure in AN-SYS Fluent form. This nodal pressure is then mapped on the CFD ANSA model using the Fluent > AUXILIARIES > RESULTS function with a scale factor of 10e-6 (*because all dimensions in Fluent are in m*) as shown in ***Figure 19*** and ***Figure 20***. The next step is the transformation of nodal pressure to NASTRAN PLOAD2 form, which can be made by using ANSA User Script Buttons > ResultsToLoads function and give the result that is shown in ***Figure 21*** and ***Figure 22****.*

Figure 19: *Nodal Pressure mapped on ANSA CFD model (wing's upper side)*
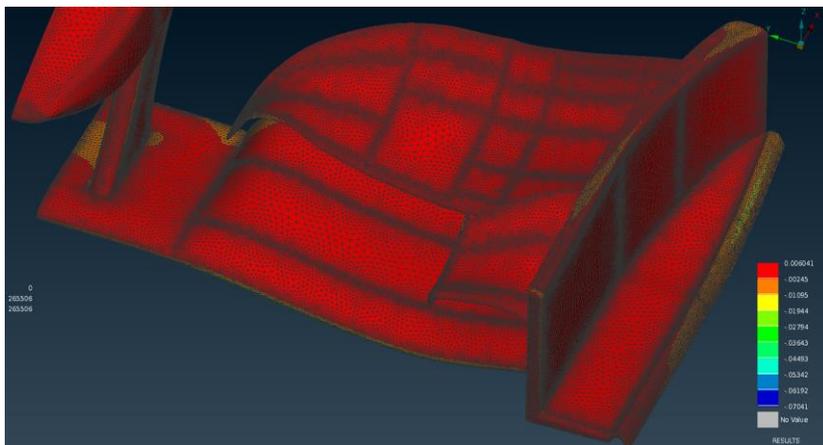
Figure 20: *Nodal Pressure mapped on ANSA CFD model (wing's downside)*
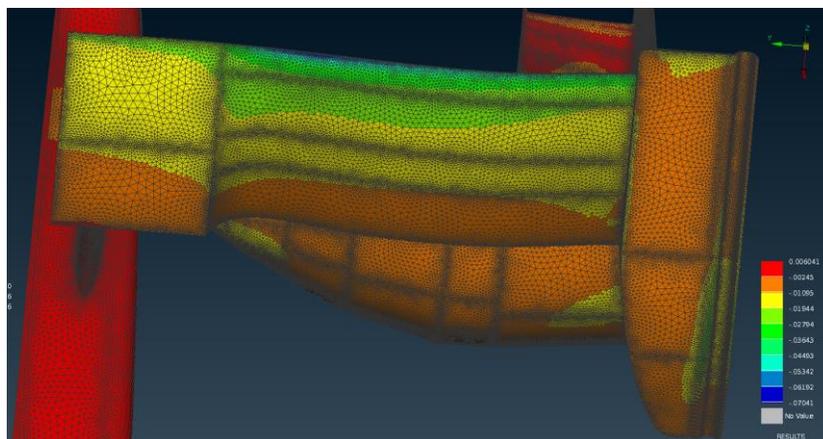
Figure 21: *Nodal Pressure translated to NAS-TRAN PLOAD2 on AN-SA CFD model*
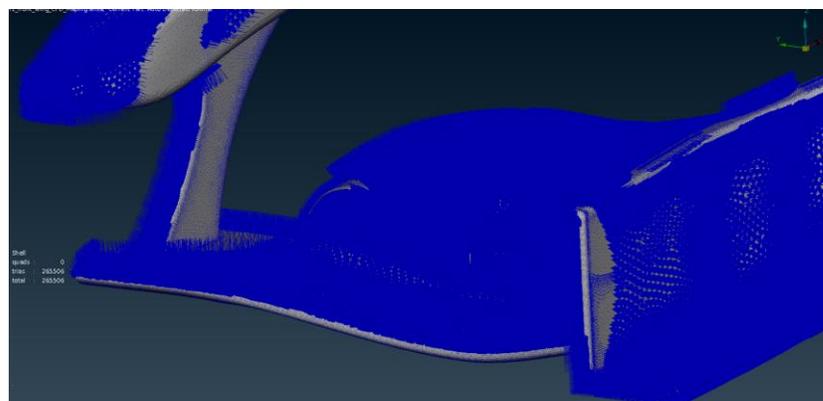
Figure 22: *NASTRAN PLOAD2 on CFD trias mesh elements*
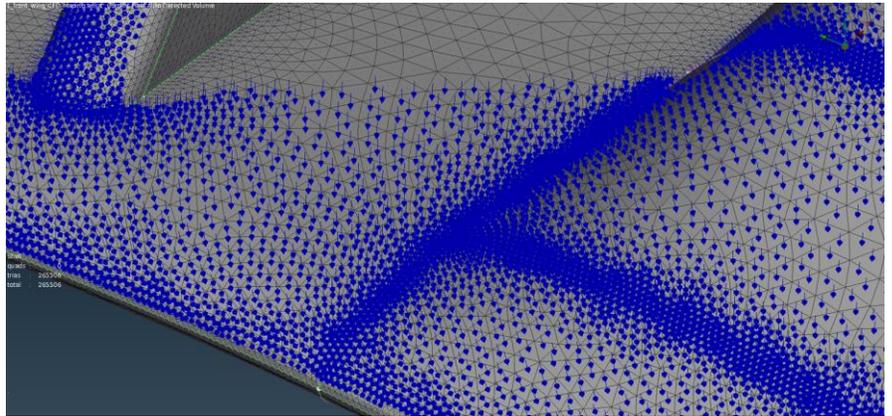
Again an output file in .nas form is created and then used to transfer the PLOAD2 to the FEA model. On the FEA ANSA model the NASTRAN > AUXILIARIES > RES.MAP function is used in order to map the PLOAD2 from the .nas file and create the pressure distribution that is needed to set *Subcase_2,* giving the result shown below on ***Figure 23*** and ***Figure 24.***The FEA model is then fully defined and ready for the optimization process to take place.

Figure 23: *NASTRAN PLOAD2 mapped on ANSA FEA model*
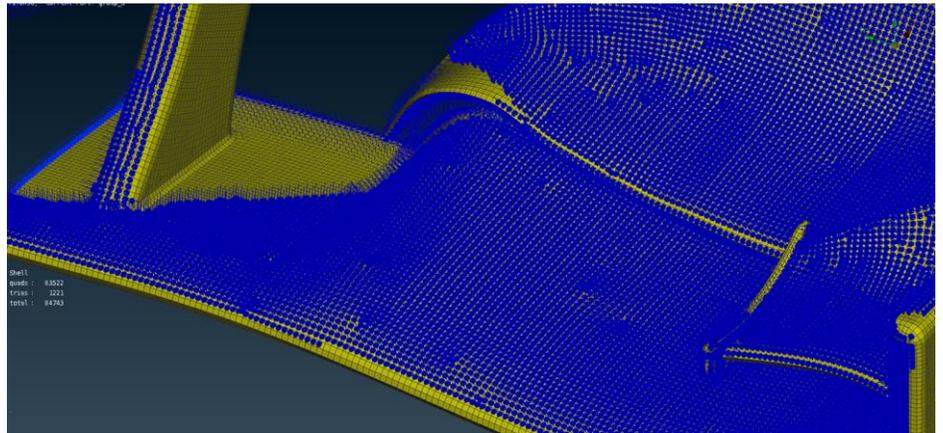
Figure 24: *NASTRAN PLOAD2 on FEA tetras mesh elements*
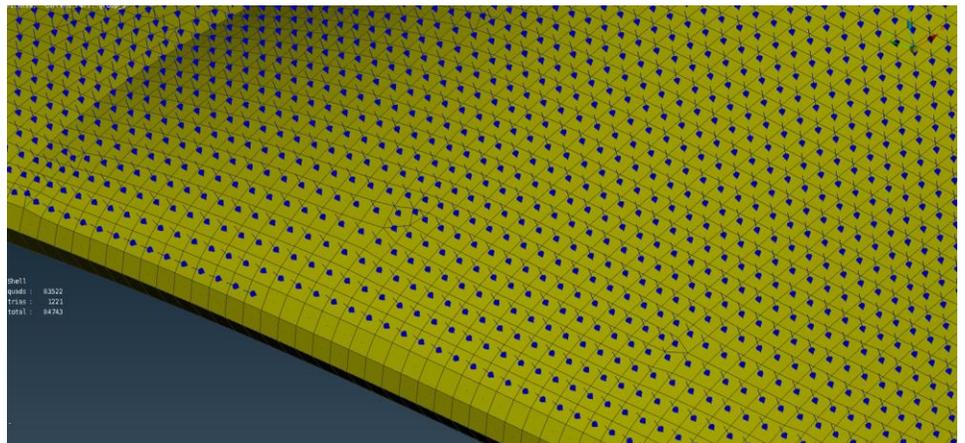
4. NASTRAN SOL200 OPTIMIZATION

4.1. Model Setup

Due to the anisotropic properties of Composites materials, it is necessary first to define an initial model with specific thickness distribution in the critical regions of the wing, in order to expedite the composite optimization process to be accurate and require less time. This optimal thickness model is going to be used as the initial model for the creation of Layers and determine the initial number of layers for each zone thickness. To achieve this the whole wing model is firstly defined with an Isotropic material (*Steel*) and next an element thickness optimization with NASTRAN-*SOL200* for both *Subcase_1* and *Subcase_2* takes place in order to retrieve an intial element thickness distribution. The FEA model is configured for NASTRAN-SOL200 Topometry optimization process (***Figure 26***) with aim the minimization of the wing's weight. Six thickness design variables are used, one for each part of the model and two constrains for Displacement (*less than 10mm*) and minimum compliance*.*

Figure 25: *Initial thickness of the wing model with Isotropic material before the optimization*
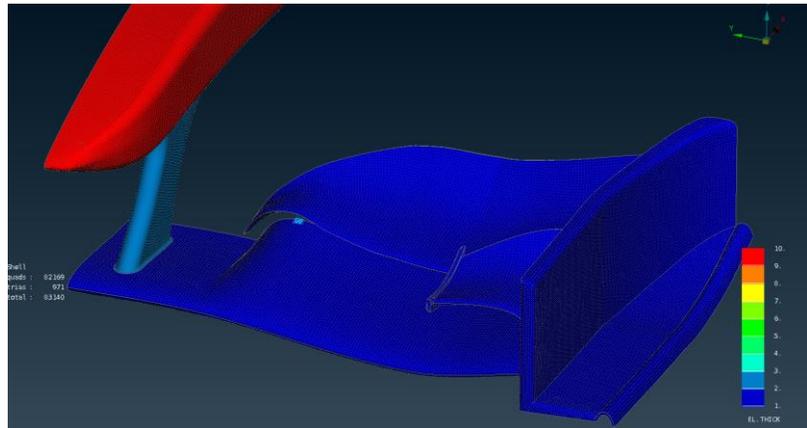
Figure 26: *Task Manager for NASTRAN-SOL200 Topometry Optimization process*
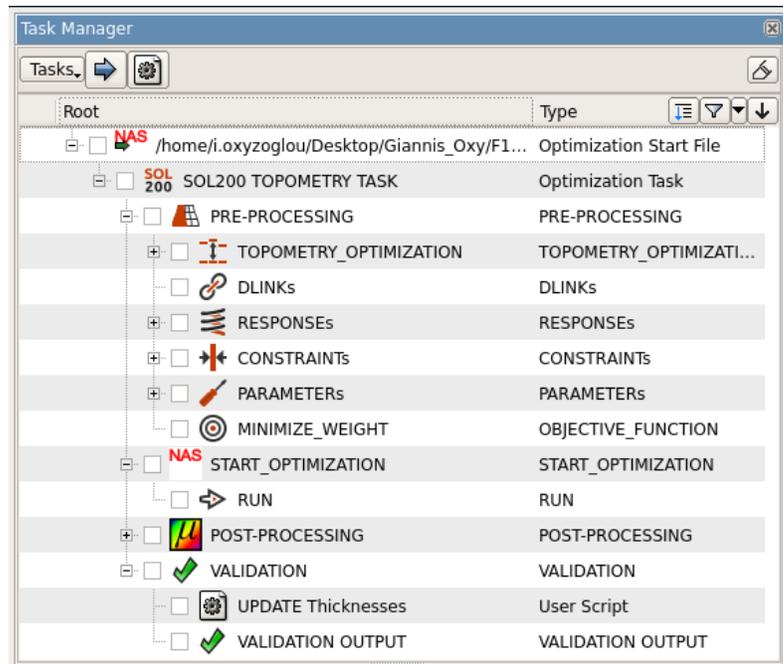
Figure 27: *Design variables, Responses and Constrains used for the element thickness optimization*

After an optimization process of 30 cycles in total the initial model has to be modified so that an optimized model with only the required element thickness results, with a displacement less than 10mm for *Subcase_1* (***Figure 29***) and minimum stresses for *Subcase_2.*

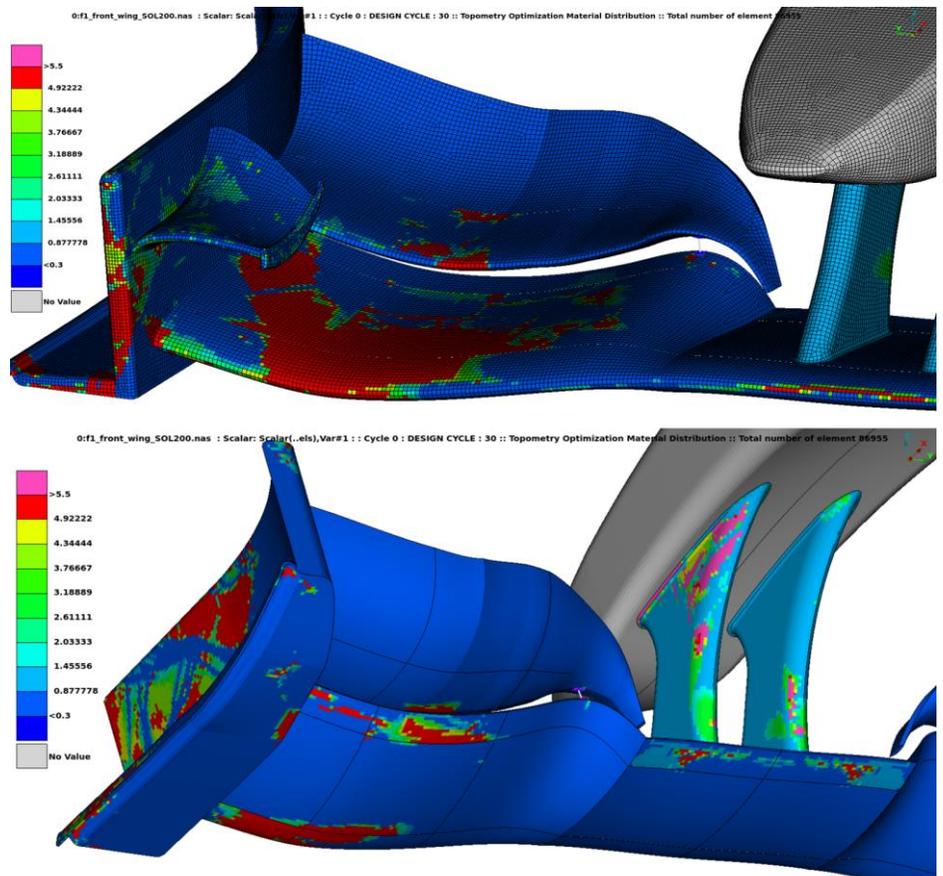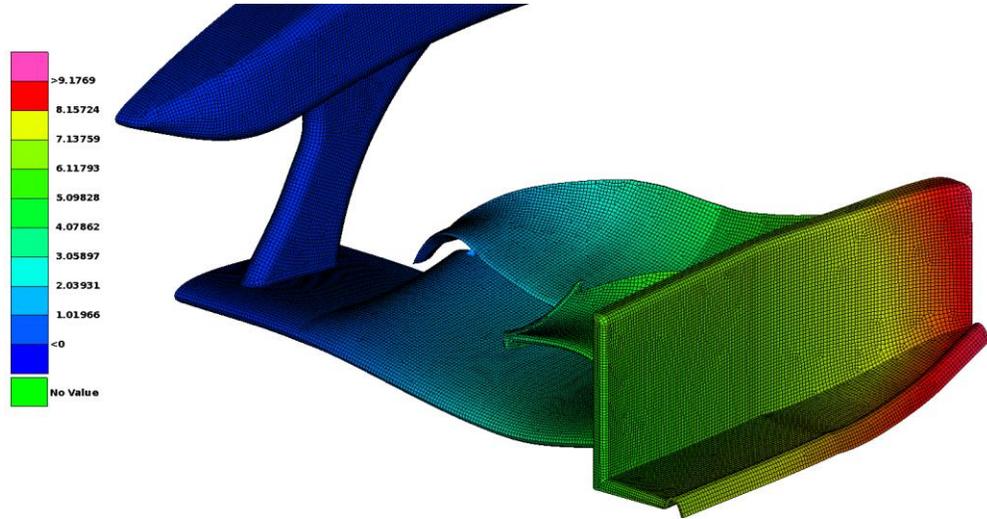**Figure 28:** *Element thickness optimized model after the NASTRAN-SOL200 optimization*

17

Figure 29: *Maximum displacement is about 9.17mm for Subcase_1*

4.2. Element Thickness Mapping

The final step before proceeding to Composite optimization is to transfer the element thickness from the optimized model to the FEA ANSA model. For this reason, an output file including the wing's optimal thickness distribution is created in mETA and used to map this thickness on the ANSA model. Using ANSA Tools > Plugins > Mapping > mETA Results and choosing the output file, the element thickness distribution transferred to the FEA model and mapped on the target parts. To simplify the Composite optimization process only the two airfoils, the endplate and the nosecone pillar are selected as target parts for optimization excluding spars, ribs and the nosecone. The final FEA model with mapped element thickness is shown in *Figure 30* below.

Figure 30: *Final Isotropic model with optimal element thickness after the NAS-TRAN-SOL200 optimization*

18
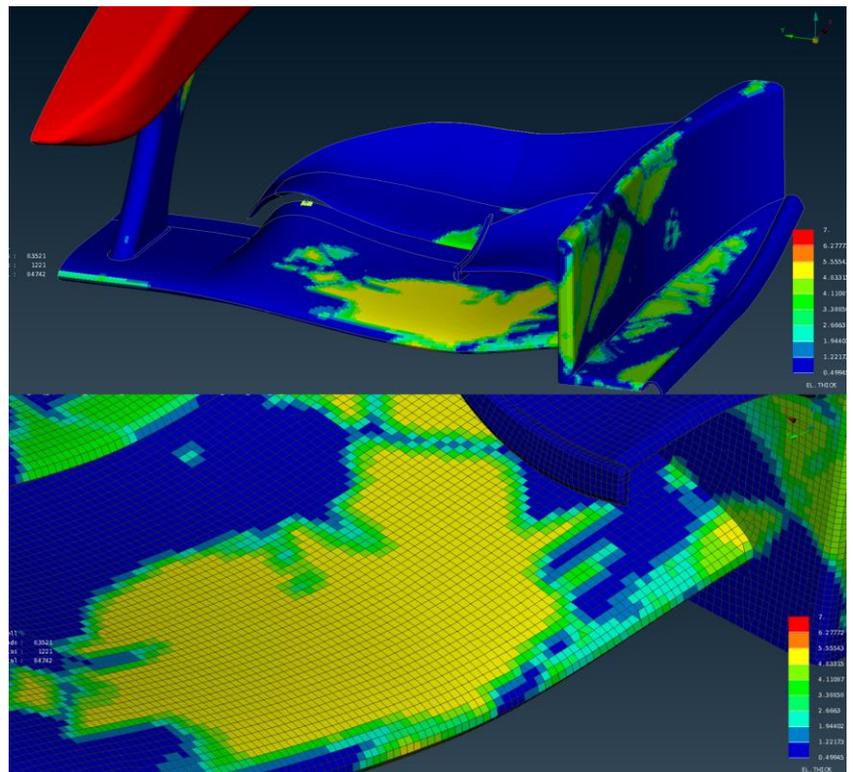
5. COMPOSITE STRUCTURE OPTIMIZATION

5.1. Element Thickness conversion to Laminate Layers

When the SOL200 thickness optimization of the Isotropic model is over the only thing left is to setup the composite model based on the final optimized thickness. However, it is necessary to use a Python script to transform the PSHELLS to Laminates, by creating a specific number of layers based on the thickness of each element that is mapped after the NASTRAN-*SOL200* optimization. The *AutoLayerFromThickness* script is used with a maximum of 26 layers for the elements with highest thickness while a small thickness smoothing took place for the nearest elements by creating less layers. This result in a laminate model consisting of 26 layers in total with random initial layer thickness and orientation. However, many of these 26 layer are scattered in different parts of the wing model so it is first necessary to split these layers to smaller layers for each part, which is going to give a model with a total of 61 layers (**Figure 31**). Finally, in order to make the simulation more accurate these 61 layers are also slightly modified to be manufacturable and drapable so that the model could be closer to a realistic experimental model as shown in **Figure 32**.

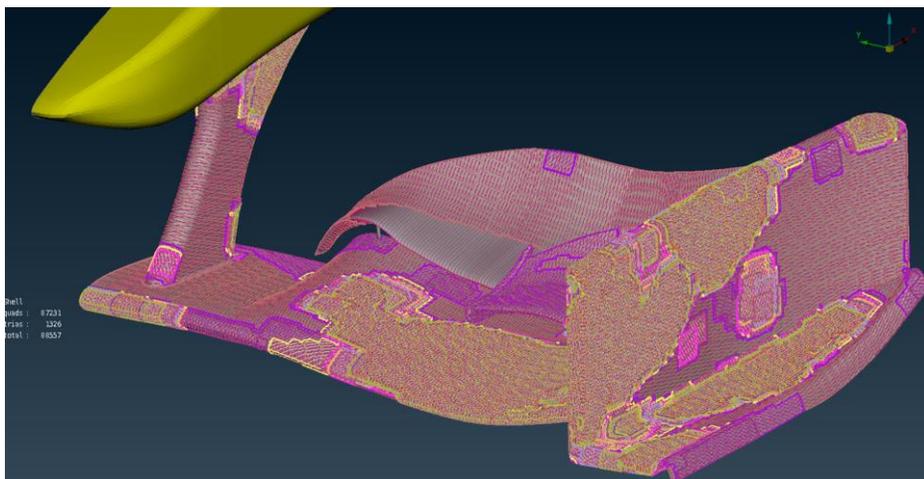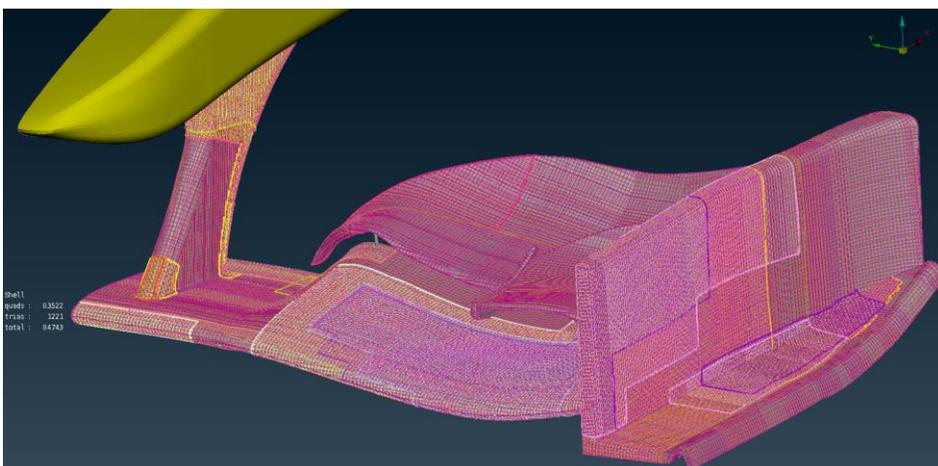**Figure 31:** *Initial 26 layers Laminate model based on SOL200 optimized thickness*

Figure 32: *Realistic model with 61 modified drapable layers separated for each part*

5.2. Layers Draping

However, in reality all elements of a layer don't have the same orientation, because during the manufacturing process a layer can be streched in order to cover properly all curvature areas, thus many elements result having different orientation than expected. To simulate this phenomenon the ANSA Draping Tool is used after any change of layer thickness or orientation during an optimization cycle. So, two Rosettes have been created on the main airfoil as starting points for the Draping, applying to all layers with a -x direction (**Figure 33**). A Python Script shown in **Figure 34** also been created and added on the Task manager in order to be executed repeatedly, draping only the modified layers that their orientation or thickness changed. However, the main airfoil consisted of both woven and unidirectional layers, so the Drape process has to be done separately with Sliding Algorithm for Unidirectional layers and Fishnet for Woven layers which is also handelded by the Script. The result of the Drape process for two layers after the execution of the script is shown in **Figure 35**.

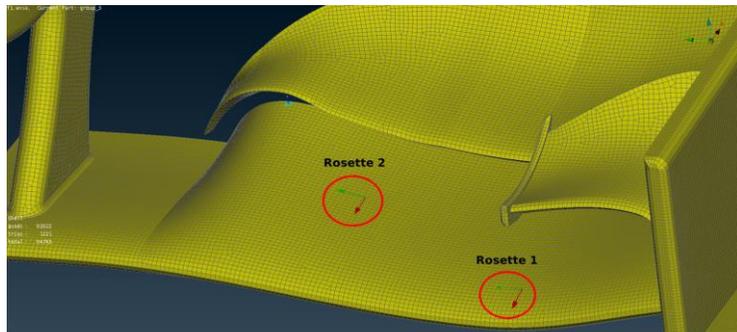**Figure 33:** *Rosettes used for Draping tool*

Figure 34: *Drape Script used to automatically Drape layers at each optimization iteration*

```python
import os
import ansa
from ansa import base
from ansa import constants

def Drape_Layers():
    laminate= base.GetEntity(constants.NASTRAN, "LAMINATE", 1021);
    sequence_UD=[]
    sequence_FB=[]
    for i in range(2, 90):
        info=base.LaminateLayerInfo(constants.NASTRAN, laminate, 1, i, {"MID"})
        if info["MID"]._id==1:
            layer_UD=(1,i)
            sequence_UD.append(layer_UD)
        else:
            layer_FB=(1,i)
            sequence_FB.append(layer_FB)
    base.SetLaminateRosette(constants.NASTRAN, laminate, 1, sequence_UD)
    base.SetLaminateRosette(constants.NASTRAN, laminate, 2, sequence_FB)
    base.DrapeLaminate(constants.NASTRAN, laminate, sequence_UD, 60, "MAXIMUM", False, 5, True, "Sliding")
    base.DrapeLaminate(constants.NASTRAN, laminate, sequence_FB, 60, "MAXIMUM", False, 5, True,"Fishnet")
```

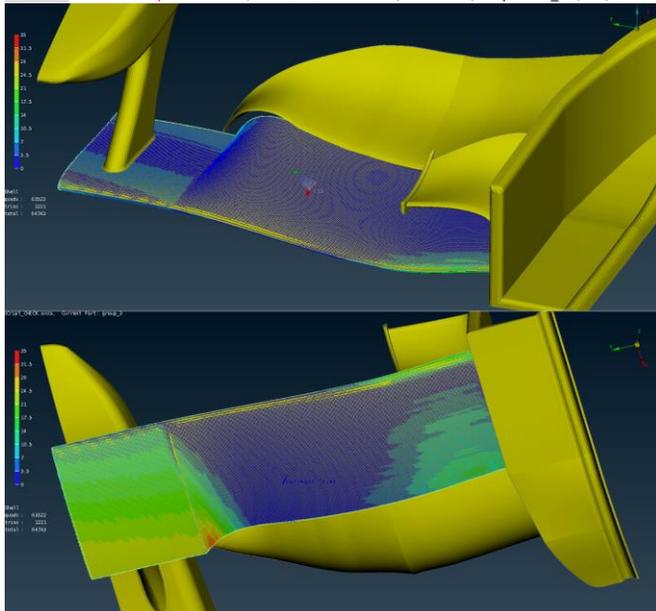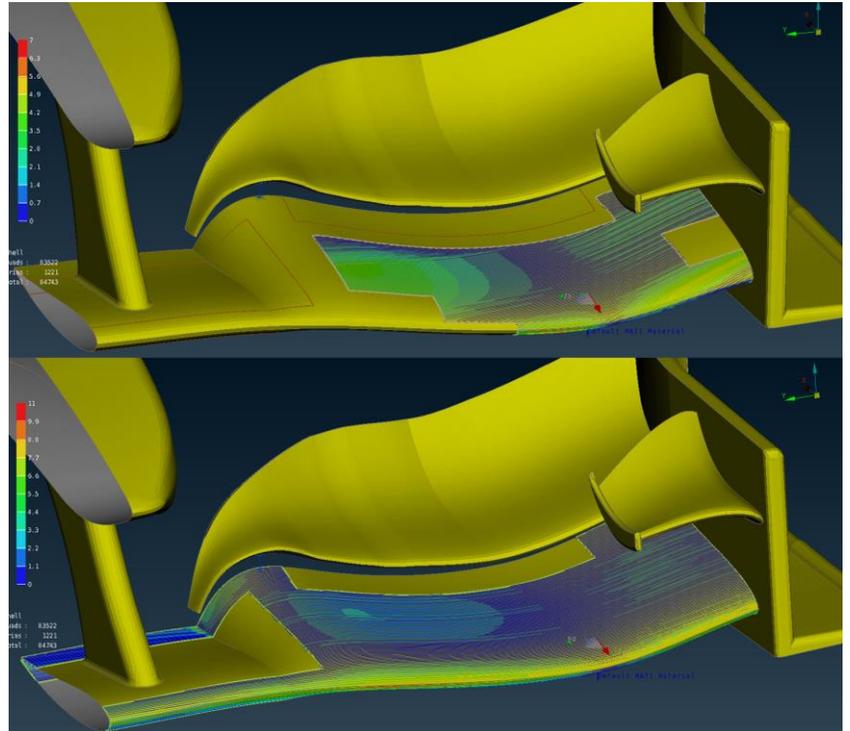**Figure 35:** *Example of a Draped Woven Layer after the execution of the Script*

Figure 36: *Example of a Draped Unidirectional Layer after the execution of the Script*

5.3. Definition of Composite Materials Properties

In order to set up the FEA model properly the correct definition of the Composite materials properties is necessary. Kevlar material is used for the outer skin layers of the wing, because of its braid which holds the whole structure and protects the tires during a crash. So, the skin layer is a woven layer with 0°/90° orientation and a thickness of *0.4mm* and doesn't participate in the optimization process because it is used only for manufacturing purposes. The skin layers that covered the whole surface of the wing on the interior are defined as high modulus carbon fibers with fabric properties at 0°/90°. The rest of the interior layers which cover only specific regions of the wing are also defined as high modulus carbon fibers, but with Unidirectional properties because with slight changes of their orientation they could greatly decrease composite stresses. Finally, the nosecone has a total of 13 layers with a single layer of Nomex Honeycomb as core in the middle but it is also excluded from the optimization process. On the ANSA model these mechanical properties are inserted on MAT8 tables for each composite material.

Table 6: *MAT8 table of High Modulus Carbon Fiber(Fabric)*

21

Table 7: *MAT8 table of High Modulus Carbon Fiber(Unidirectional)*

MAT8 MATT8 MATEP MATF [MAT8]

Name: Unidiractional Carbon Fiber

FROZEN_ID: NO   FROZEN_DELETE: NO   DEFINED: YES

MID	E1	E2	NU12	G12	G1Z	G2Z	RHO
1	250000.	6500.	0.31	3200.	3200.	3200.	1.65E-9

A1	A2	TREF	Xt	Xc	Yt	Yc	S
			1580.	744.	22.	210.	61.

GE	F12	STRN

MATT8: NO

Unidiractional Carbon Fiber

OK   ColorEdit   Cancel

Table 8: *MAT8 table of Nomex Honeycomb*

MAT8 MATT8 MATEP MATF [MAT8]

Name: Nomex Honeycomb

FROZEN_ID: NO   FROZEN_DELETE: NO   DEFINED: YES

MID	E1	E2	NU12	G12	G1Z	G2Z	RHO
5	75.84	75.84	0.1	25.	25.	25.	3.2E-11

A1	A2	TREF	Xt	Xc	Yt	Yc	S
			1034.	1034.	1034.	1034.	1.

GE	F12	STRN

MATT8: NO

Nomex Honeycomb

OK   ColorEdit   Cancel

Table 9: *MAT8 table of Kevlar 49*

MAT8 MATT8 MATEP MATF [MAT8]

Name: Kevlar 49 Fabric

FROZEN_ID: NO   FROZEN_DELETE: NO   DEFINED: YES

MID	E1	E2	NU12	G12	G1Z	G2Z	RHO
4	30000.	30000.	0.2	5000.	5000.	5000.	1.4E-9

A1	A2	TREF	Xt	Xc	Yt	Yc	S
7.4	7.4		480.	190.	480.	190.	50.

GE	F12	STRN

MATT8: NO

Kevlar 49 Fabric

OK   ColorEdit   Cancel

22

5.4. Task Manager Setup

In order for the optimization process to begin the FEA model must be compatible with *modeFRONTIER*. A user Optimization Task is settled on the Task manager with an output DV_file including all the design variables with their min and max values, a Session Command with Deck Info for the total mass of the wing which is also the target variable of the optimization and the .edb output file for the EPILYSIS solver. Layer thickness and orientation are considered as design variables of the Optimization Task with angles between 45° and -90° and a step value of 15°. The initial thickness for each layer of the 61 layers is settled as 1.4 mm which is obviously an unrealistic scenario, but considering a step value of 0.2mm which is the manufacturing thickness of a composite layer, the optimized model could result in a total layer thickness multiple of 0.2 which could be separated later to an integer number of layers with 0.2mm thickness.

Table 10: *Task Manager for the modeFRONTIER Optimization*
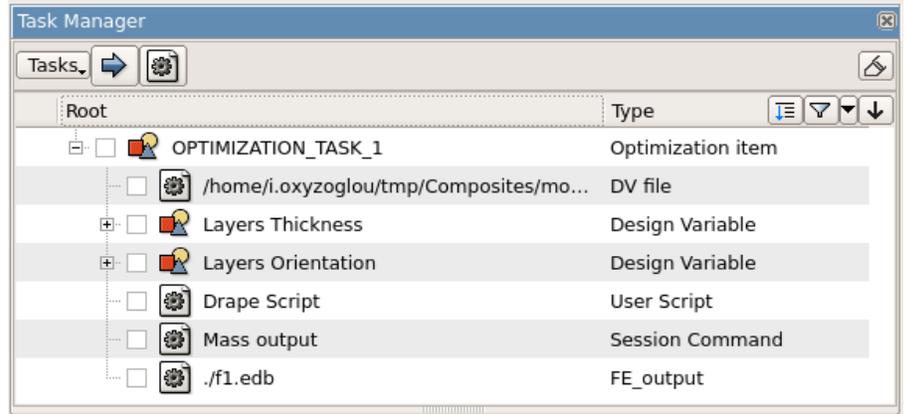
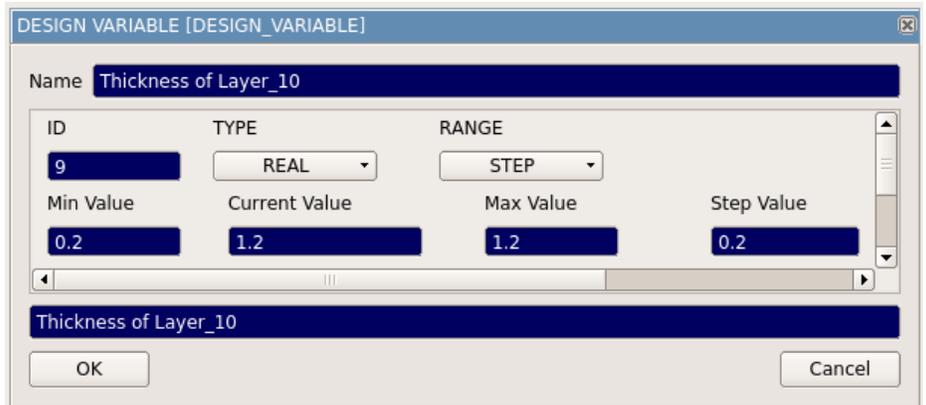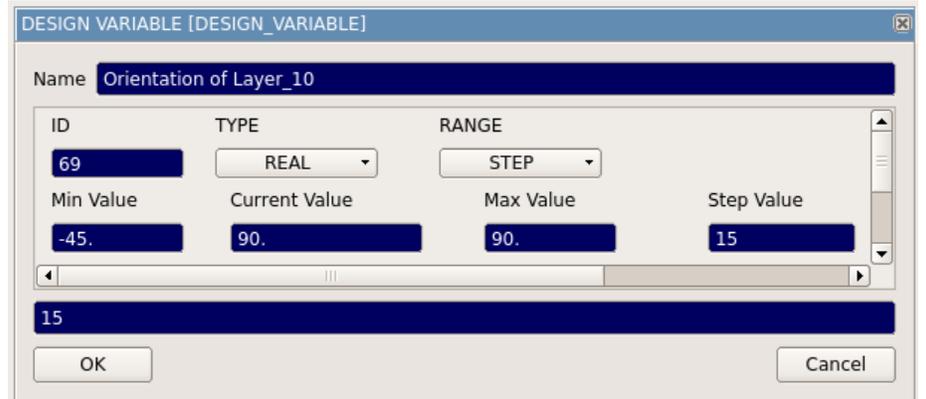**Table 11:** *Design variable Table for Layer thickness*

Table 12: *Design variable Table for Layer Orientation*

23

5.5. Definition of the Constrains

Lastly the constrains for the Optimizer have to be defined. By solving the initial Composite model with EPILISYS and importing results to mETA *post*-Processor the two constrains for each *Subcase_1* and *Subcase_2* can be easily defined. As already mentioned the constrain for *Subcase_1* is the total Displacement of the wing's lowest point not to be greater than 10mm and is defined by creating an Annotation with max Element Scalar value picking the lowest element at the lowest edge of the wing (**Figure 37**). The aim on *Subcase_2* is the reduction of the composite stresses, so based on the Tsai-Wu criterion which says that if the minimum Reserve Factor of all layers is lower than 1 then failure occurs and by using a safety factor of 3 the constrain is defined as the minimum Reserve Factor of all Layers being greater than 3. Again, after the visualization of the minimum Reserve Factor of all layers, an Annotation is created for all the visible elements, except the nosecone's (**Figure 38**). Finally, a Respones.ses output file is created in mETA containing all the constrains and later imported to modeFRONTIER.
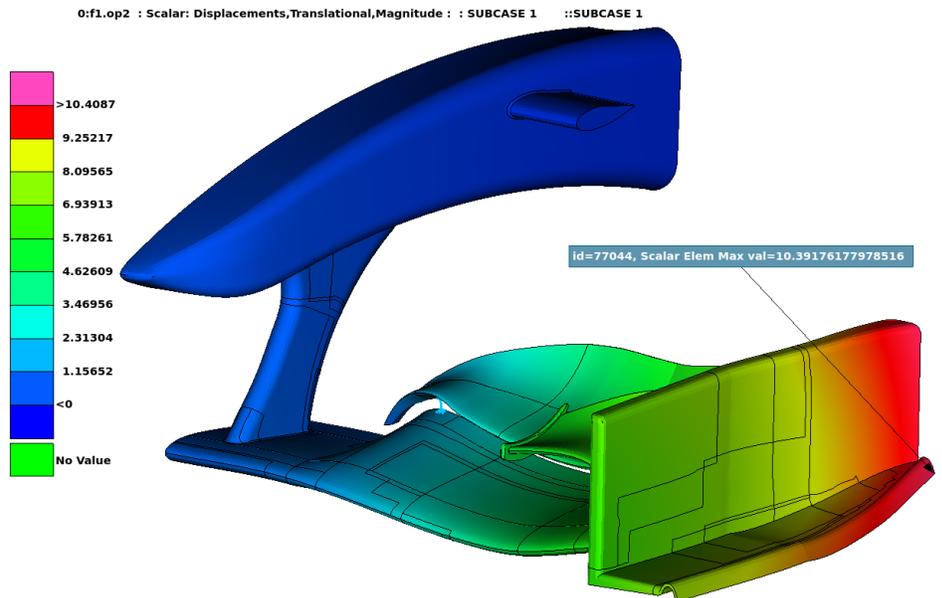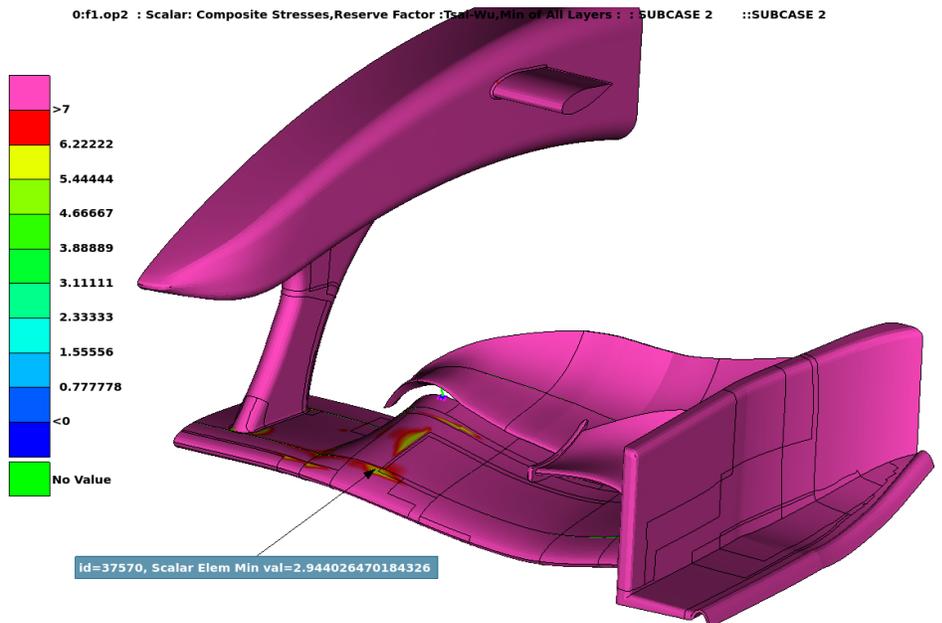
Figure 37: *Displacement constrain for Subcase_1*

Figure 38: *Minimum Tsai-Wu Reserve Factor constrain for Subcase_1*

5.6. ModeFRONTIER Setup

Using *modeFRONTIER* an iterative process is settled aiming to the reduction of the wing's weight while minimizing at the same time the total Displacement and keeping the minimum Reserve Factor of all layers above 3. To do so, a procedure chart is used, connecting actually *ANSA*, *EPILYSIS*, *mETA* and the optimizer together, creating essentially a complete Optimization process. At each iteration, the design variables layer thickness and orientation are changing based on their step values creating a new *ANSA* model which is later being solved with *EPILYSIS* until the results on *mETA* be compatible with the constrains and the mass reach its minimum feasible value. The Multi Objective Genetic Algorithm (*MOGA-II*) designed for fast Pareto convergence selected while it uses directional crossover operator for fast convergence and elitism to ensure that the best solutions are preserved during the evolution. The optimization has been done using an i7 Desktop Computer with a RAM memory of 128GB and the CPU time needed is
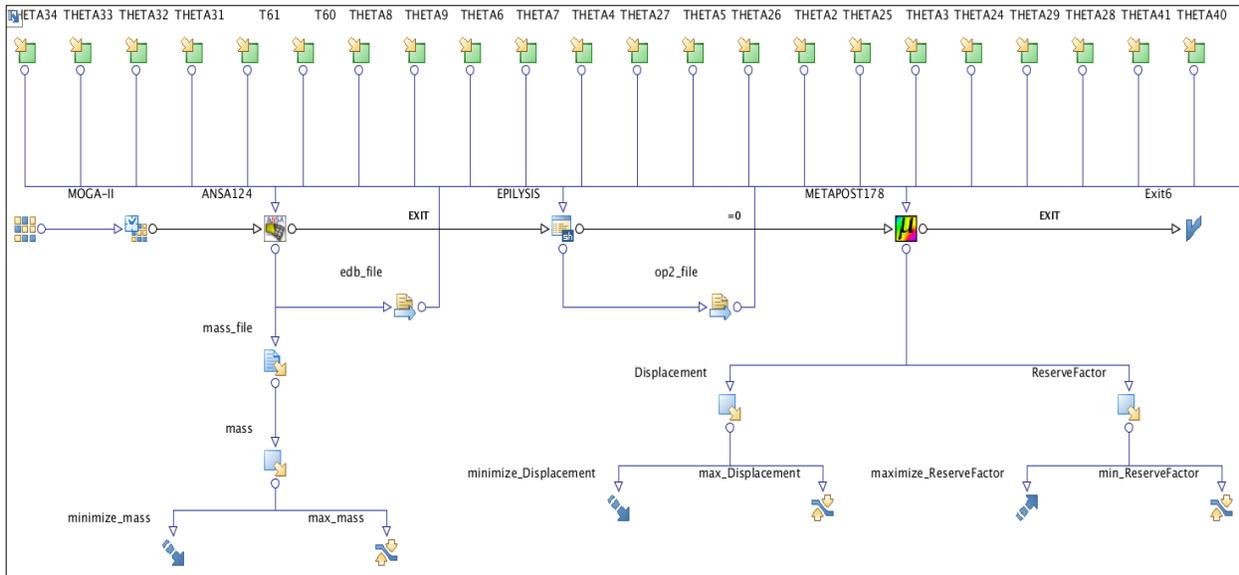
about 7 days.

Table 13: *Displacement constrain as settled in modeFRONTIER*

Table 14: *Minimum Reserve Factor constrain as settled in modeFRONTIER*

Figure 39: *Optimization Procedure diagram settled in modeFRONTIER*

25

5.7. Optimization Steps & Resutls

Due to the large number of the design variables the Optimization process is separated in 4 basic steps in order to minimize total time required and better control the convergence of the algorithm. On the first step the initial model of the 61 layers with 1.4mm thickness each and total mass of *16.965kg* is optimized by changing at the same time both its layer thickness and orientation, resulting in a model with a total mass of *11.845kg.* The total number of layers is still 61 but the thickness of each layer is now a multiple of 0.2 which can easily be divided in an integer number of layers with 0.2mm thickness each. After the separation of the 61 layers a model of 314 layers occured, however many of the new layers that earlier belonged to the same layer, they now have the same orientation which is the main reason that the Displacement doesn't reach below the 10mm constrain on the 1st Step. The next three steps have to do with the optimization of each part of the wing one by one until the total mass reach a satisfactory low value and complying with all constrains.
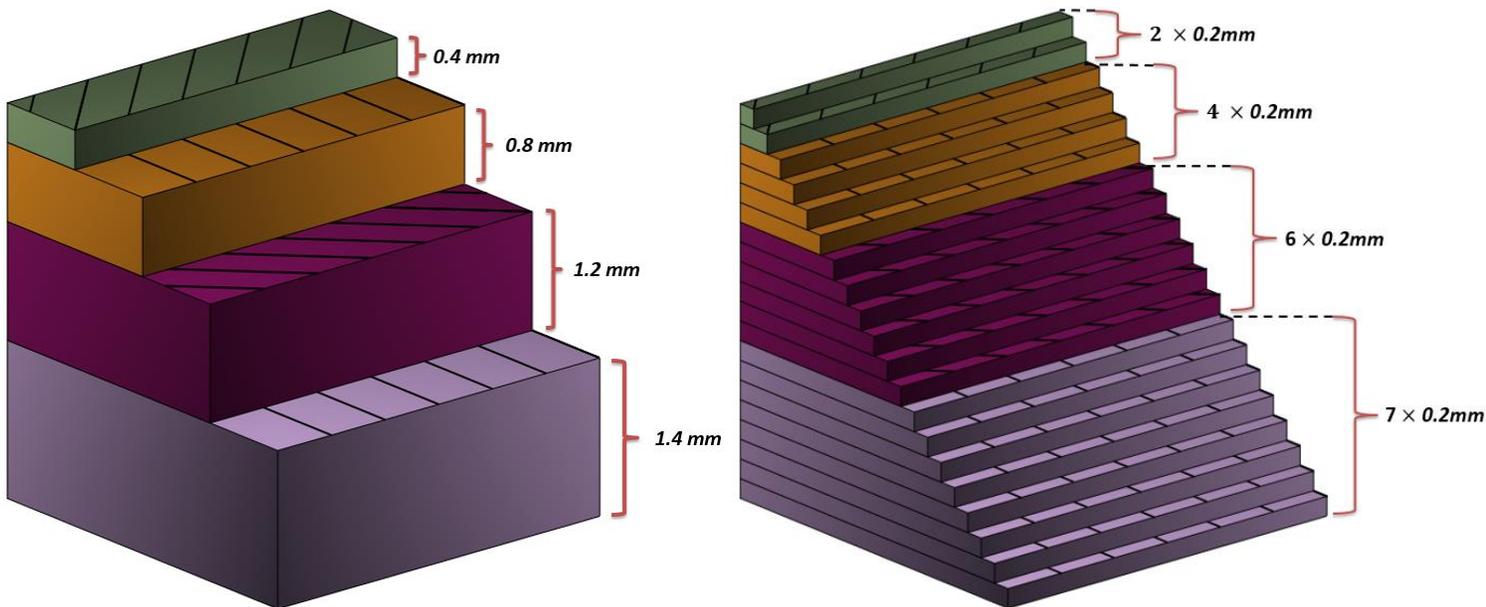

Figure 40: *Initial batch of 61 layers splitted to 314 layes of 0.2mm Thickness each*

After splitting the 61 initial batches of layers the number of the design variables is 628 (*314 for orientation & 314 for thickness*) so, these three steps have to be further separated in three sub-steps starting with an orientation optimization, followed by a thickness optimization and then again by an orientation optimization. After the execution of all these steps a final model of 207 occured with a total mass of 8.221kg, a total Displacement of 9.956mm and a Reserve Factor of 3.017. The only reason that the Optimization is chosen to be treated this way is to further restrict the infinite combinations of solutions and fit it in a feasible time period because due to the enormous number of the design variables if the Optimization is done at once for all the parts with both thickness and orientation changing, it may require even more time to reach an optimized solution. However, for models with less number of layers or if more CPU power is available and by using a more accurate optimizing algorithm which treats the whole process differently, the user may choose to make a direct approach of the optimization process combining all steps in one.

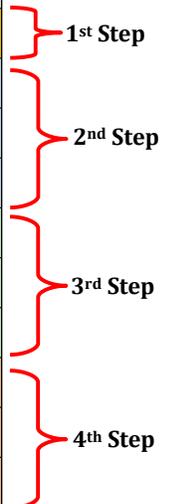| OPTIMIZATION PROCESS					
CASES	**CONSTRAINS**				
	Mass (*kg*)	**Displacement (*mm*)**	**Reserve Factor**	**# of Layers**	
Initial Wing (*based on SOL200*)	16.965	9.715	3.27	61	
All parts Thickness & Orientation	11.845	10.233	3.024	61	1st Step
Pillar 1st Orientation	11.845	9.379	3.146	314	
Pillar Thickness	11.402	9.507	2.936	250	2nd Step
Pillar 2nd Orientation	11.402	9.589	2.651	250	
Endplate & 2nd Airfoil 1st Orientation	11.402	9.582	2.624	250	
Endplate & 2nd Airfoil Thickness	8.543	9.889	2.603	211	3rd Step
Endplate & 2nd Airfoil 2nd Orientation	8.546	9.784	2.552	211	
1st Airfoil 1st Orientation	8.546	9.958	3.125	211	
1st Airfoil Thickness	8.471	10.206	3.023	209	4th Step
1st Airfoil 2nd Orientation	8.471	9.961	3.245	209	

Table 15: *Optimization Process results after each step*

27

Chart 2: *Mass variation during the Optimization process*

Chart 3: *Step to step Reserve Factor variation until to be compatible with Tsai-Wu criterion and safety factor 3*
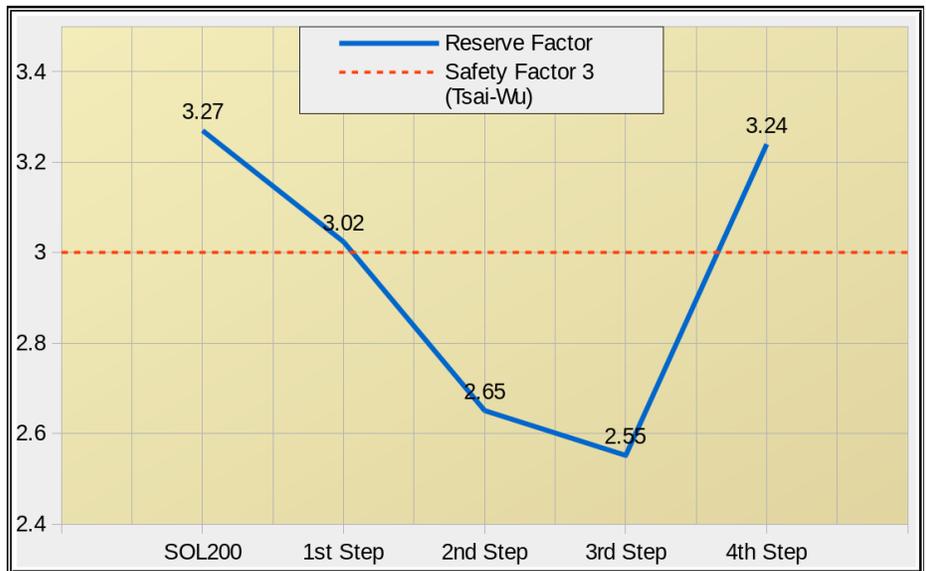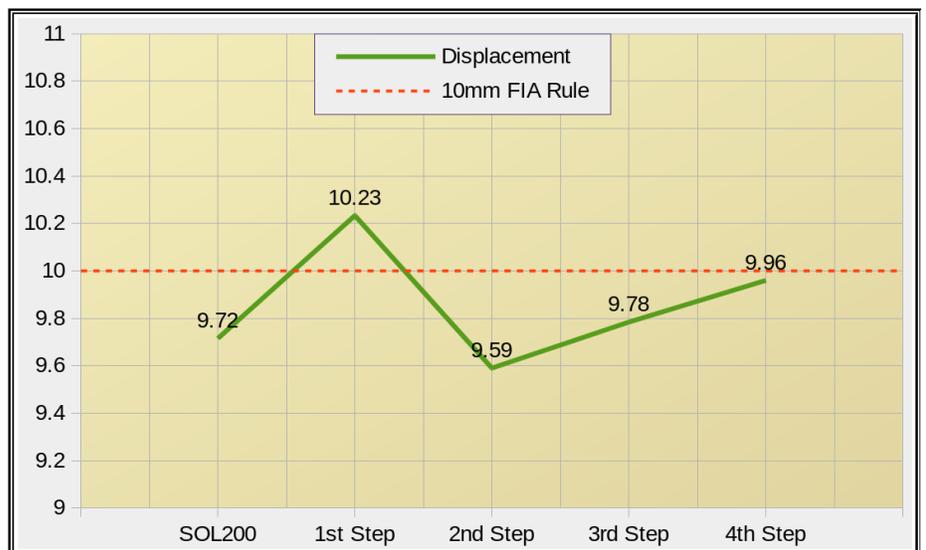

Chart 4: *Step to step Displacement variation until FIAs 10mm Rule confirmed*

28

6. RESULT ANALYSIS

6.1. Models Comparison

After the successful completion of the Optimization process an extensive analysis of the final results has to be done. In order to estimate the accuracy of the results and emphasize the overall benefits gained from this optimization process a comparison between the optimized model and an initial simple layer model has to be made. The simple model for comparison shown in **Figure 41** has been created by using whole layers covering all the surfaces of the wing, with the same material properties that are used in the optimized model. The total number of layers used in order to satisfy the same constrains is 41, with a thickness of 0.2mm each and a typical orientation (*based on composites theory*) of 0°/45°/0°/-45°/90° which resulted in a total mass of *12.117kg.* The total Displacement of this model is 9.983mm (**Figure 42**), while the minimum Reserve Factor of all Layers is 3.757 (**Figure 43**).

Figure 41: *Initial wing model, with 41 layers covering on all the surfaces and 12.117kg of mass*
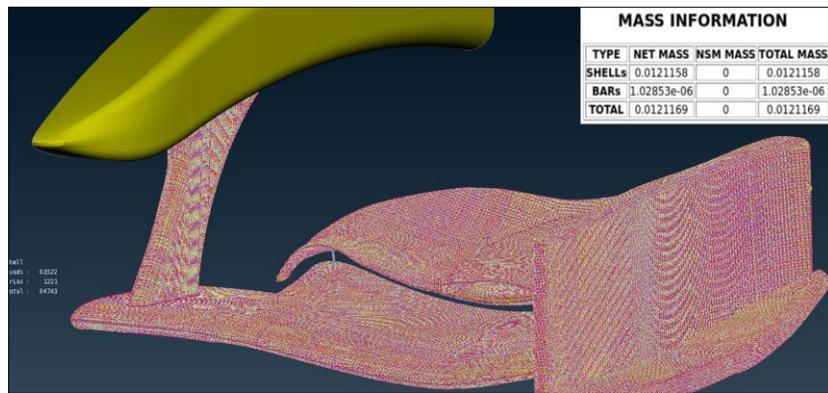

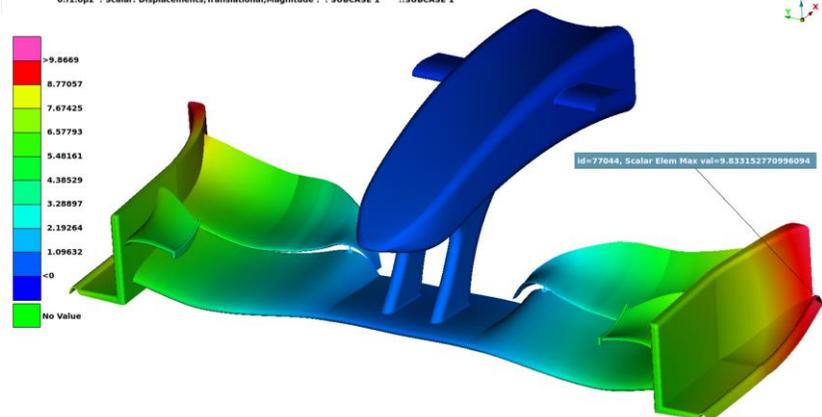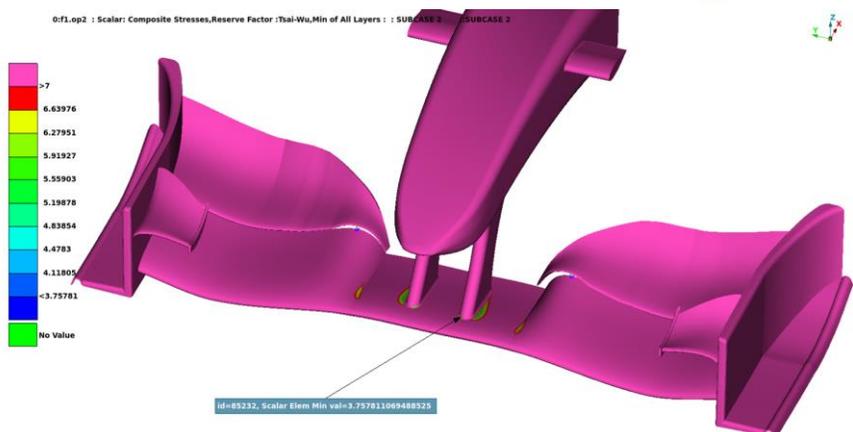**Figure 42:** *Total Displacement of the simple model is 9.83mm*

Figure 43: *Minimum Reserve Factor of all layers for the simple model is 3.757*

29

At the end of the 4th step of the Optimization the optimized wing model shown in *Figure 44* is produced. This model consists of 209 layers in total with 0.2mm layer thickness each and a total estimated mass of 8.471kg. Although the mass of the wing is reduced by 30% comparing with the initial model, the total Displacement is 9.961mm (*Figure 45*), being still compatible with FIAs rules while the minimum Reserve Factor of all Layers is 3.245 (*Figure 46*), marginally greater than the safety factor 3 that is settled as constrain.

Figure 44: *Final optimized wing model, with 209 layers and 8.471kg of mass*
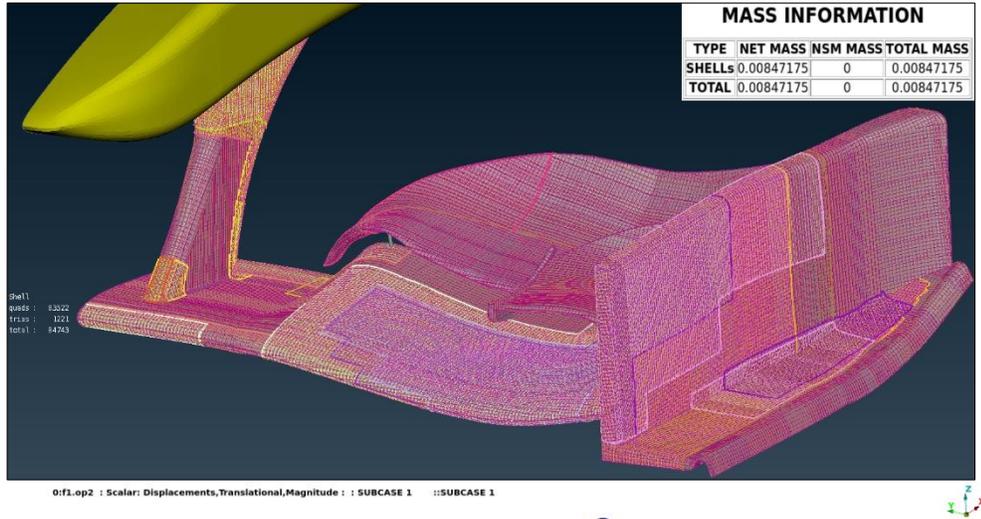

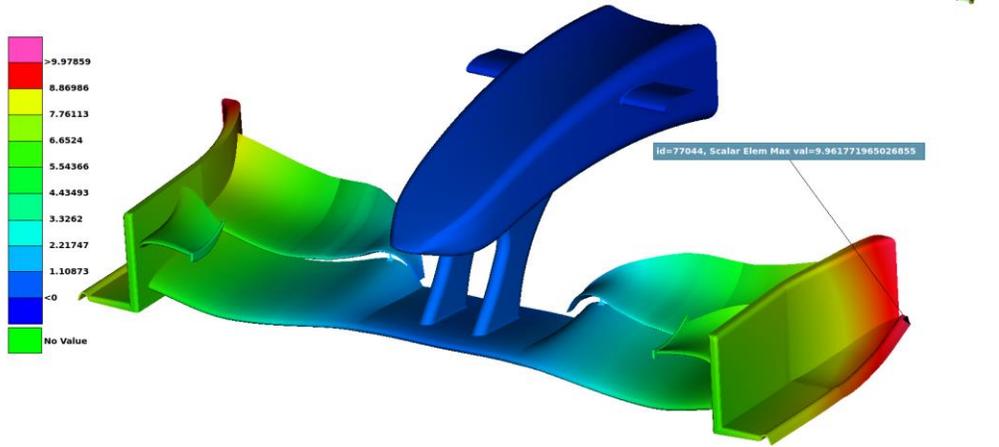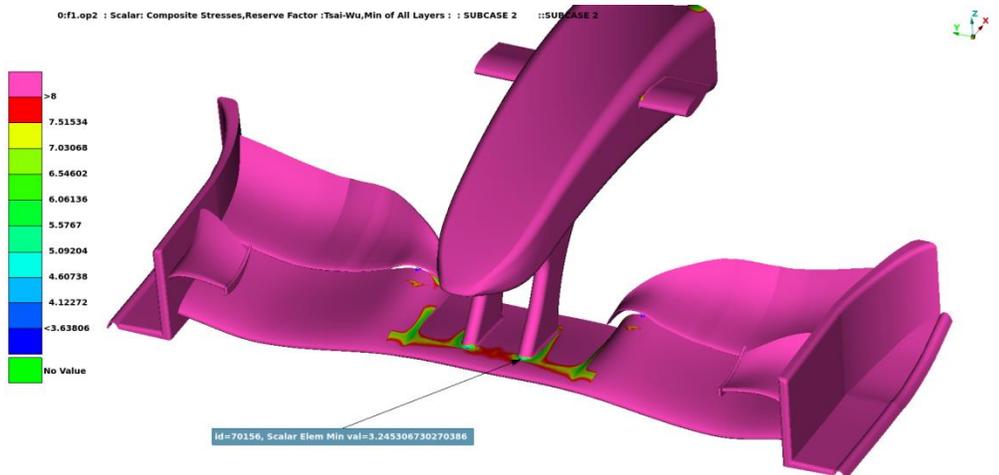**Figure 45:** *Total Displacement of the optimized wing is 9.961mm*

Figure 46: *Minimum Reserve Factor of all layers for the optimized model is 3.245*
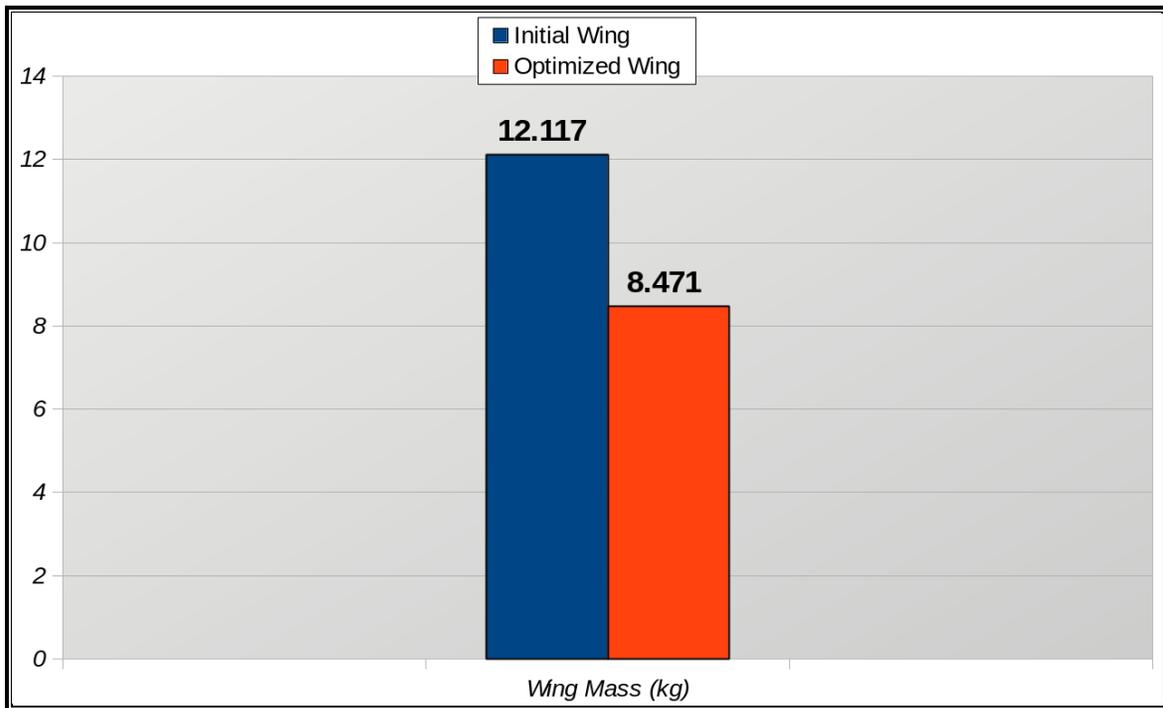
6.2. Weight & Cost Analysis

Chart 5: *The optimized wing occured about 3.65kg lighter than the initial wing, achieving a mass decreased up to 30%*
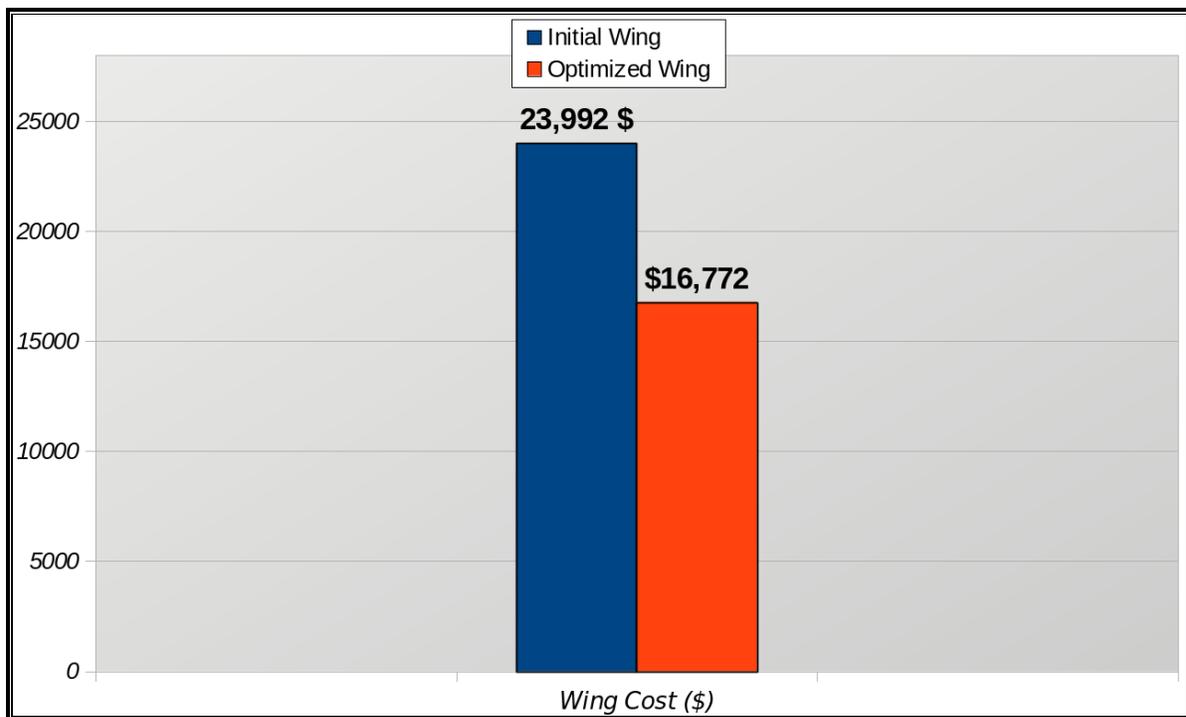

Chart 6: *Considering that the material cost of High-Modulus Carbon Fibers is about 1980$/kg, the cost of the optimized wing is 7,220$ less than the initial wing's (30% reduction)*
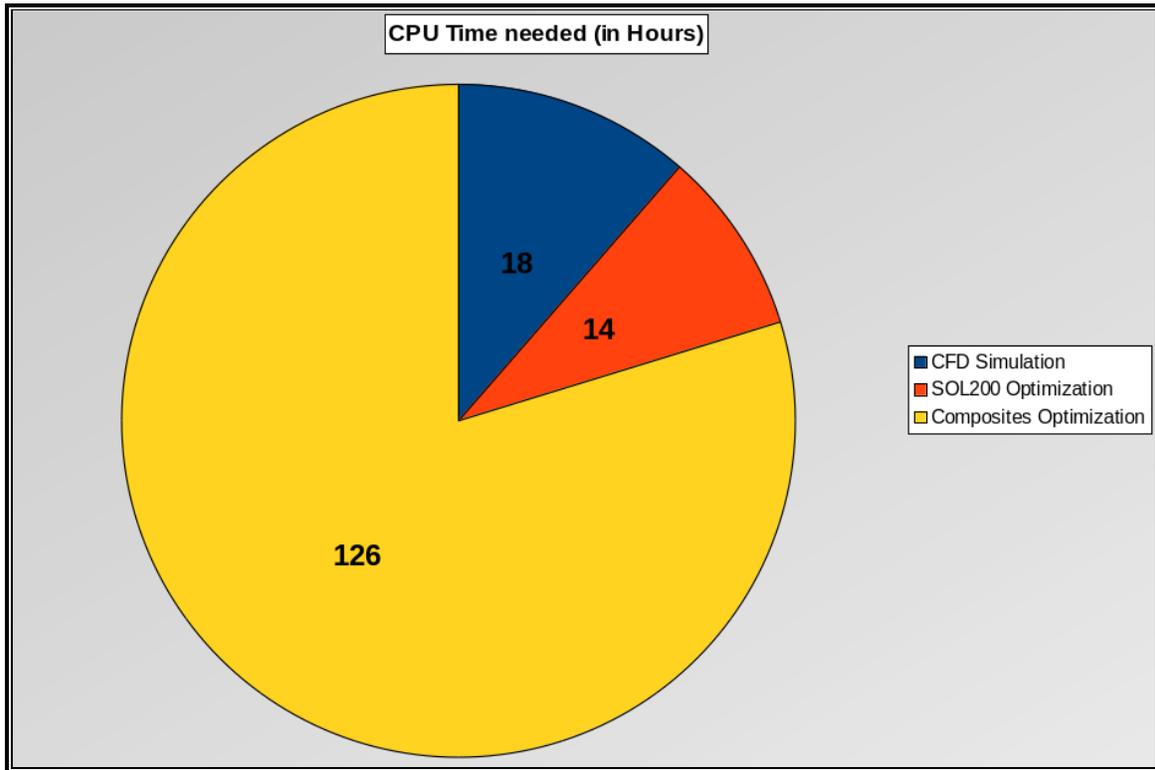
6.3. Total Process Time

Chart 7: *Total CPU time (in hours) required for the completion of each process of the Optimization. By using precise algorithms specializing in composite optimization and with more CPU power available the composite optimization time can be significantly reduced*

7. BIBLIOGRAPHY

i.   Tsai, S. W., & Melo, J. D. (2015). ***Composite Materials Design and Testing.***

ii.   Gary Savage, Honda Racing F1. (2008). ***Composite Materials Technology in Formula 1 Motor Racing.***

iii.   Lazaros Tsioraklidis, Altair Engineering. (2013). ***Cost Optimization of a composite F1 front wing under aerodynamic loading.***

iv.   ***Red Bull Racing Flexible front wings Season 2011.*** (n.d.). *retrieved from* Formula 1 Dictionary.

v.   ***Carbon Fiber reinforced Composite Materials.*** (n.d.). *retrieved from* Formula 1 Dictionary.

vi.   ***Mechanical Properties of Carbon Fiber Composite Materials, Fiber / Epoxy resin (120°C Cure).*** (n.d.). *retrieved from* ACP Composites.

vii.   ***Mechanical Properties of Carbon Fibre Composite Materials, Fibre / Epoxy resin (120°C Cure).*** (n.d.). *retrieved from* Performance Composites: Composite Manufacturing.

viii.   ***Tsai–Wu failure criterion.*** (2016). *retrieved from* Wikipedia.

ix.   McDaniel, K. (2011). ***Optimization with non-Homogeneous Failure Criteria like Tsai-Wu for Composite Laminates.***