iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 14 **Issue:** IV **Month of publication:** April 2026

DOI: https://doi.org/10.22214/ijraset.2026.79149

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 14 Issue IV Apr 2026- Available at www.ijraset.com

Modeling and CFD Analysis of Formula 1 Front Wings

Sanket Khopade, Chinmay Pawar, Yash Dhanave, Kaustubh Landge, Prof. N.R. Pawar
DEPARTMENT OF MECHANICAL ENGINEERING, PADMABHOOSHAN VASANTDADA PATIL INSTITUTE OF TECHNOLOGY, India

Abstract: The CFD is very much useful, and it is an appropriate tool for the flows type of problems. Whenever the object dynamic analysis to be carried out there involves the computation of the flow properties and analysis based on the movement and object orientation. The configuration of the F1 vehicle wings is done using ANSYS software. Using the ANSYS software the modeling and analysis of computational fluid dynamics is performed. As it's a known thing that F1 cars have the high speed to move on the roads. Hence thedrag and lift forces are emphasized for understanding. The wings inF1are an important element presentin the F1 cars for its movement inthe proper direction. Analysis of the rear wing gives a complete picture of key parameters and areas to be focused and they are to be understood with respect to the betterment of the performance.

Keywords: Modeling,CFDanalysis,Formula1frontwings,ANSYS

I. INTRODUCTION

Formula One, also known as Formula One or F1, and currently officially referred to as the FIA Formula One World Championship, is the highest class of auto racing organized by the Fédération Internationale de l'Automobile (FIA). The "formula" in the name refers to a set of rules that all participants and cars must follow. Aerodynamics has become key to success in the Formula One sport and spends millions of dollars on research and development in the field each year. The aerodynamic design has two primary concerns [1]. First, creating a down force to help push the car's tires onto the trackand improve the cornering force, secondly, to minimizing the drag caused by turbulence and act to slow the car down. To create the down force to the car, the wing operates with airflow at different speeds over the two sides of the wing by travelling different distances over its contour and form. This creates a pressure difference. This pressure can make the wing tries to move in the direction of the low pressure. One car capable of developing cornering force by three and a half timesits Itight produces from aerodynamic down force [1]. That means, theoretically, at high speed, the car could drive upside down.AFormulaOnecarisasingle-seat,open-cockpit, open-wheelracingcar withsubstantialfrontandrearwings, and an engine positioned behind the driver, intended to be used in competition at Formula Oneracing events. The regulations governingthecars areuniquetothechampionshipandspecifythatcars mustbeconstructedbyther acing teams themselves, though the design and manufacture can be outsourced [2].

The aerodynamic designer has two primary concerns: the creation of down force, to help push the car's tyres onto the track and improve cornering forces; and minimizing the drag that gets caused by turbulence and acts to slow the car down. Several teams started to experiment with the now familiar wings in the late 1960s. Race car wings operate on the same principle as aircraftwings but are configured to cause a downward force rather than an upward one. A modern FormulaOne car can develop 6 Gs of lateral cornering forcethanks to aerodynamic down force [3]. The aerodynamic down force allowing this is typically greater than the weight of the car. That means that, theoretically, at high speeds, they could drive on the upside-down surface of a suitable structure, e.g., on theceiling. All cars have the engine located between the driver and the rear axle [4]. The engines are a stressed member inall cars, meaningthat the engine is part of the structural support framework; being bolted to the cockpit at the front end, and transmission and rear suspension bolted at the back end of the engine [5]. Modern F1 car has more than 5,000 different parts and wind tunnels work around the clock, testing newversionsofacar,butthereare anawfullotofotherareas whicharecompletelyinvisibleto thepeopleinthe grandstands, or those watching the racing on television.

II. LITERATURE SURVEY

Aerodynamic research in F1 has been an area of high investment in the past 30 years.Assuming no regulatory limitations, this trend would continue while the bodywork rules are changed continually or while changes to the shape of the cars continue to provide significant improvements in lap time around the F1 circuits of the world.

HoItver, agreements between teams started to limit investment and now rules have been introduced officially to limit how much research is done into aerodynamics inwind tunnels and inCFD.Naturally teams will optimize their resources to still obtain the maximum they can from the aerodynamics of the cars [6].

To investigate the aerodynamics of a F1 car the teams use various methods of research. Tests are conducted using scale models of the cars in a wind tunnel fitted with a “rolling road”. Computers are used to mathematically simulate the flow of air around and through the cars and to model vehicle behaviour on the track. The real cars used to be tested as Itll in wind tunnels and onspecial straight line test facilities, but this is nowbanned [7]. Cars are tested onthe real tracks ofcourse, but that too is limited to fewer test days than was possible in the past.

Eachofthe top 10 F1teamshassomewhere between50 and 150 people workingsolelyon aerodynamic research.Itisquite difficult to be sure about how many people work on aerodynamics as generally the teams don’t talk about it. Despite the limitsontracktestingandonaerodynamicresearchthathavebeenimposedbythe rulemakers(theFIA)andby agreements between the team, the amount of research done is “significant”.Even a “small” team of 50 people can do a lotofwork!Teamsusecomputersimulationmoreandmoretopredictperformanceandtoanalysemany“whatif” scenarios.Thesesimulationsarethenconstantlyimprovedbycomparingthemtotherealitiesofracingandtesting [8]. From these tools It know that the drag of the cars does slow them down quite a lot but typically, on an average racetrack, it slows them by about 3% - 5% in lap time.In other words, if drag were to be reduced to zero, the gain in lap time at a typical track would normally be a bit less than 5%.However, if we remove the present levels of down force then lap times get slower by about 25% or so.

In the present F1 environment, other performance factors which are normally very important have been limited more severely.E.g., the tiresare allsupplied byone supplier and are carefullyrandomlyselected for the teamsbythe rule makers and tire supplier together so there is no chance of one team getting an advantage [9].The tire supplier selects 2 of the 4 dry options they make for the season plus one intermediate and one extreme wet tire.So you cannot develop your own tire togetanadvantage.Inthe 5 yearsto 2014 engine specifications were, effectively, frozen.For 2014 there isa completely new power train formulabut the idea is that these power trains will also be virtually frozen once a reasonable level of parity is established.Cars must race above a certain minimum weight (to protect against dangerous construction as low Itight helps lap time).Suspension kinematics is relatively free so this is an area where the teams can make a difference, but suspension must be passive. However, because aerodynamics is so dominant, even this is compromised to ensure aerodynamicsbenefits are maximized [10]. CFD is coming into its own as far as racing car aerodynamics are concerned.Modern super computers allow the use of mathematical models that mean complete and reasonably realistic full-vehicle aerodynamic simulationsarenowpossible,ifalittleslow.TheteamshavenowmainlysettledonaCFDmethodcalled Navier Stokes which copes Itll with the realities of racing cars.Some teams are combining the use of commercially available packages with in-house computer programs/enhancements to maximise the gains that can be made using the computers.It will be some time before it is possible to dispense with wind tunnel testing because wind tunnels allow us to very quickly test hundreds of combinations of conditions and vehicle attitudes [11].

Computational fluiddynamics (CFD)isoneofthebranches offluid mechanicsthatusesnumerical methodsandalgorithms to solve and analyse problems that involve fluid flows. Computers are used to performthe millions of calculations required to simulate the interactionofliquidsand gases withsurfaces defined byboundaryconditions. The mainadvantagesofusing CFD software is that the results are obtained without the construction of the required prototype and this is very important because it can reduces the cost in constructing the F1 cars [12]. The validity of the results is the most important things that It need to concern about while using the software simulation. Therefore, the specific parameters and conditions while analysing the data need to be valid.

Lift is the components ofthe pressure and wall shear force inthe directionnormal to the flowtend to move the bodyin that direction. It can prevent the object from fly to the air when use the negative lift coefficient. The pressure difference between the top and bottom surface of the wing generate an upward force that tends the wing to lift. For the slender bodies such as wings,theshear forceactsnearlyparalleltothe flowdirection,thusitscontributiontotheliftissmall.Thelift forcedepend onthe density, ρ, ofthe fluid, the upstreamvelocityV, the size, shape, and orientationofthe body, amongother things, and it is not practical to list these force for a variety of situations. Instead, it is found convenient to work with appropriate dimensionless numbers that present the drag and lift characteristics of the body. These numbers are the lift coefficient, C*R*L*R*.It is defined as

$$F_L = C_L / 2\rho V^2 \qquad (1)$$

WhereAisordinarilythefrontalarea(theareaprojectedonaplanenormaltothedirectionofflow)ofthebody.½ρV^{2}is thedynamicpressureandF*R*L*R*isliftforce.

Drag is the aerodynamic force that is opposite to the velocity of an object moving through air or any other fluid.

Its size is proportional to the speed differential between air and the solid object. Drag comes in various forms, one of them being friction drag which is the result of the friction of the solid molecules against air molecules in their boundary layer. Friction and its drag depend on the fluid and the solid properties. A smooth surface of the solid for example produces less skin friction compared to a rough one. For the fluid, the friction varies along with its viscosity and the relative magnitude of the viscous forces to the motion of the flow, expressed as the Reynolds number. Along the solid surface, a boundary layer of low energy flow is generated, and the magnitude of the skin friction depends on conditions in the boundary layer.

The amount of drag that a certain object generates in airflow is quantified in a drag coefficient. This coefficient expresses theratio ofthedrag forceto theforceproduced bythedynamicpressuretimesthearea. Therefore, aC*R*d*R*of1 denotesthatall air flowing onto the object will be stopped, while a theoretical 0 is a perfectlyclean air stream. At relatively high speeds of high Reynolds number (Re > 1000), the aerodynamic drag force can be calculated by this formula

$$F_D = C_d/2\rho V^2 \qquad (2)$$

Where,

F_D=Force of drag ρ=Densityoftheair

V^2= Speed oftheobject relativetothefluid(m/s)

A=Referencesurfacearea C_d=Drag of coefficient

III. MODELLING OF FORMULA 1 WING

Computer geometric modellingis the mathematical representation of an object's geometry using software. A geometric model contains description of the modelled object's shape. Since geometric shapes are described by surfaces, curves are used to construct them. Computer geometric modelling uses curves to control the object's surfaces as they are easy to manipulate.

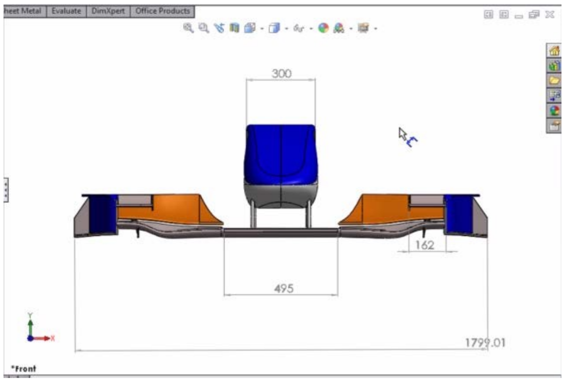


Fig.1.DimensionsofF1front winginfrontview

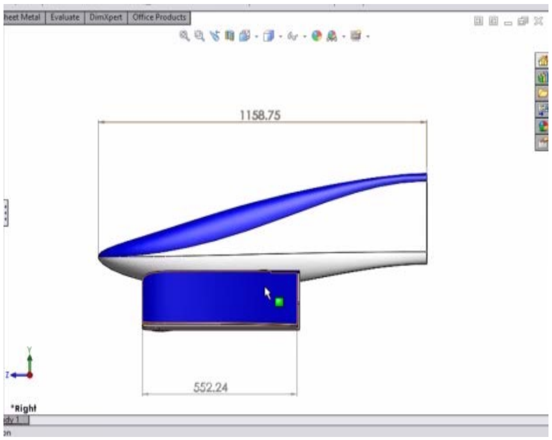
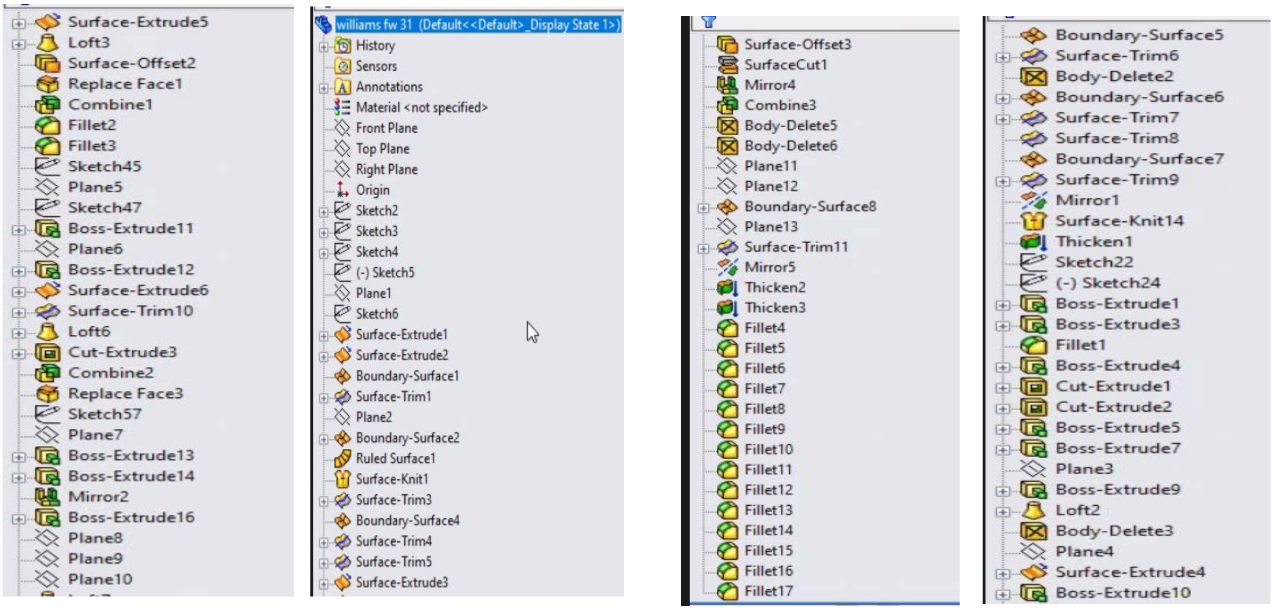


Fig.2.DimensionsofF1front winginside view



Surface-Extrude5
Loft3
Surface-Offset2
Replace Face1
Combine1
Fillet2
Fillet3
Sketch45
Plane5
Sketch47
Boss-Extrude11
Plane6
Boss-Extrude12
Surface-Extrude6
Surface-Trim10
Loft6
Cut-Extrude3
Combine2
Replace Face3
Sketch57
Plane7
Boss-Extrude13
Boss-Extrude14
Mirror2
Boss-Extrude16
Plane8
Plane9
Plane10

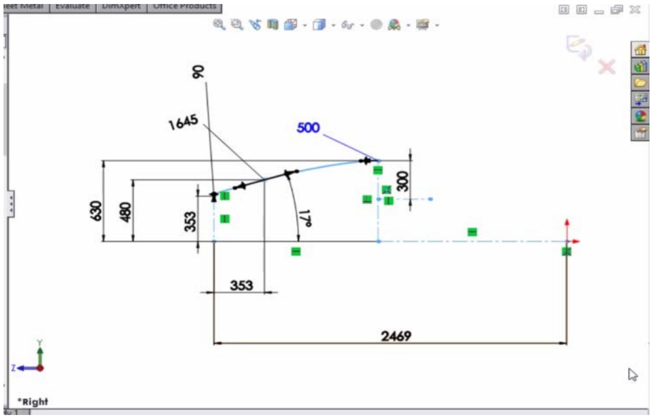
williams fw 31 (Default<<Default>_Display State 1>)
History
Sensors
Annotations
Material <not specified>
Front Plane
Top Plane
Right Plane
Origin
Sketch2
Sketch3
Sketch4
(-) Sketch5
Plane1
Sketch6
Surface-Extrude1
Surface-Extrude2
Boundary-Surface1
Surface-Trim1
Plane2
Boundary-Surface2
Ruled Surface1
Surface-Knit1
Surface-Trim3
Boundary-Surface4
Surface-Trim4
Surface-Trim5
Surface-Extrude3

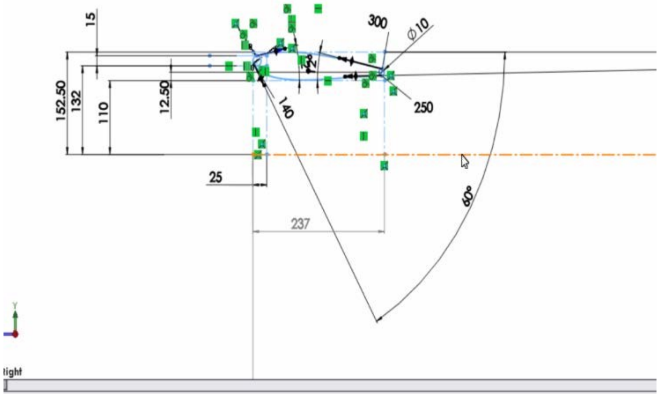
Surface-Offset3
SurfaceCut1
Mirror4
Combine3
Body-Delete5
Body-Delete6
Plane11
Plane12
Boundary-Surface8
Plane13
Surface-Trim11
Mirror5
Thicken2
Thicken3
Fillet4
Fillet5
Fillet6
Fillet7
Fillet8
Fillet9
Fillet10
Fillet11
Fillet12
Fillet13
Fillet14
Fillet15
Fillet16
Fillet17

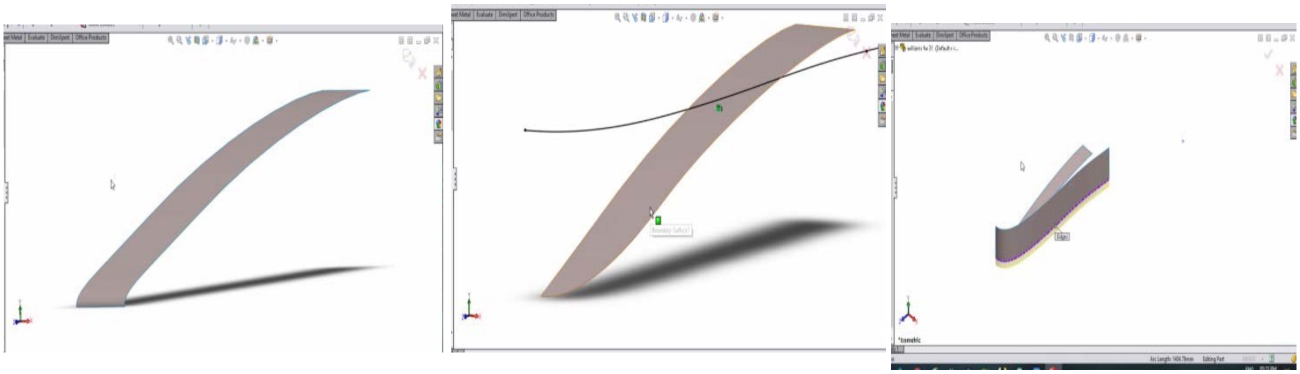
Boundary-Surface5
Surface-Trim6
Body-Delete2
Boundary-Surface6
Surface-Trim7
Surface-Trim8
Boundary-Surface7
Surface-Trim9
Mirror1
Surface-Knit14
Thicken1
Sketch22
(-) Sketch24
Boss-Extrude1
Boss-Extrude3
Fillet1
Boss-Extrude4
Cut-Extrude1
Cut-Extrude2
Boss-Extrude5
Boss-Extrude7
Plane3
Boss-Extrude9
Loft2
Body-Delete3
Plane4
Surface-Extrude4
Boss-Extrude10

Fig.3.FeaturemanagementdesigntreeofF1frontwing

- ThetreeshowsthedifferentprocessinvolvedincreatingtheF1FRONTWINGinsolid works.
- Startingfromsimplesketchestillthe3Dmodellingof F1FRONT WING.
- Fewoftheprocesses involvedin creationof3Dmodel.







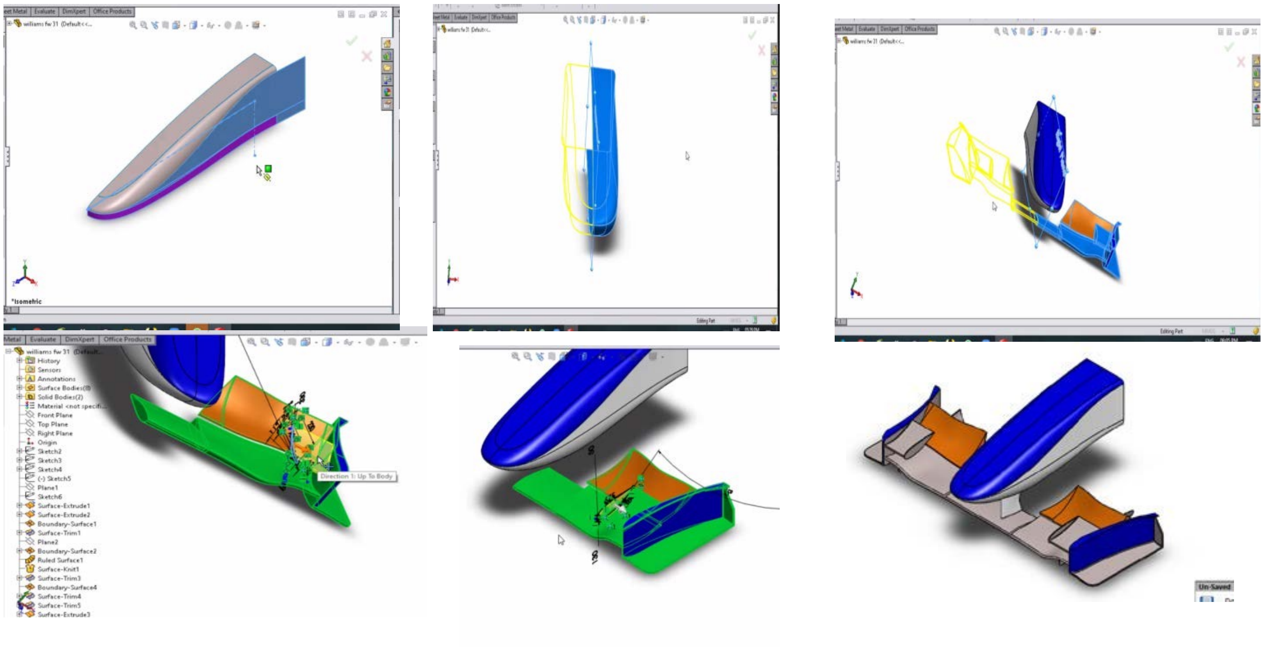


Fig.4.ModellingofF1frontwing

IV. CFD ANALYSIS OF A F1 FRONTWING

Therearefivemainstepsinvolvedforanalysisof“F1FrontWING”.Theyare.

- Geometry
- Mesh
- Setup
- Solution
- Results

TheprimarystepingeometryistoimporttheModelofafrontwingfromSolidworksusingIEGSformatandgenerate.

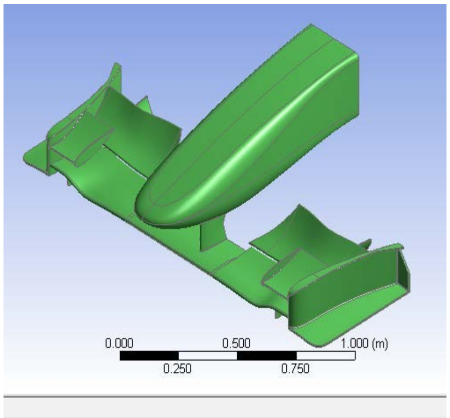


Fig.5.Importedmodelofafrontwing

Then create an enclosure from the tools tab with a uniform length of 1 meter. Last step in geometry is to create Boolean withsubtractoperation. Meshingisanintegralpartofthe computer-aided engineering(CAE) simulationprocess. Themesh influences the accuracy, convergence, and speed of the solution. Furthermore, the time it takes to create a mesh model is often a significant portion of the time it takes to get results froma CAE solution. Therefore, the better and more automated the meshing tools, the better the solution.

From easy, automatic meshing, to a highly crafted mesh, Ansys, provides the ultimate solution. Powerful automation capabilities ease the initial meshing of a new geometrybykeying off physics preferences and using smart defaults so that a mesh can be obtained upon first try. Additionally, users are able to update immediately to a parameter change, making the handoff from CAD to CAE seamless and aiding in up-front design.

Once the best design is found, meshing technologies from Ansys provide the flexibility to produce meshes that range in complexity from a pure hex mesh to highly detailed hybrid meshes; users can put the right mesh in the right place and ensure thata simulation will accurately validate the physical model..

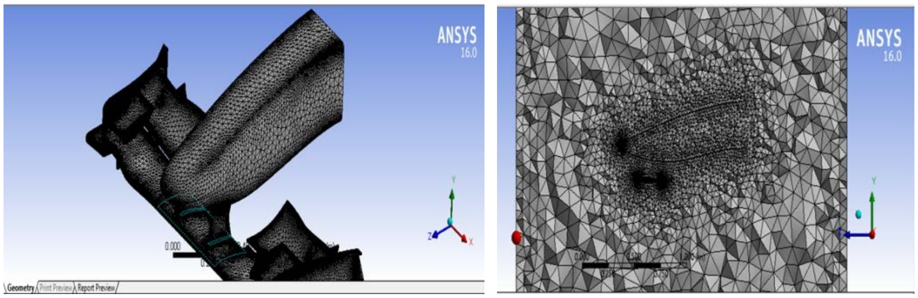


Fig.6.MeshingofF1Frontwing

Now that you have created a computational mesh for the “F1 FRONT WING” geometry, you can proceed to setting up a CFD analysis using ANSYS FLUENT.

StartANSYSFLUENT.

IntheANSYSWorkbenchProjectSchematic,double-clicktheSetupcellintheF1FRONTWINGfluidflowanalysis system. You can also right-click on the Setup cell to display the context menu where you can select the Edit option

When ANSYS FLUENT is first started, FLUENT Launcher is displayed, allowing you to view and/or set certain ANSYS FLUENT start-up options.

ANSYSFLUENTLauncherallowsyoutodecidewhichversionofANSYSFLUENTyouwilluse,basedonyour geometry and on your processing capabilities.

InANSYSFLUENT Dimensionsettingisalreadyfilledinandcannotbechanged,sinceitautomaticallysetsitbasedon the mesh or geometry for the current system.

InANSYSfluentreading, writingandbuildingofmesh,domainetc.,takesplaceautomaticallyforfluentsolver.Meshis displayed on the graphics window.

- Boundary conditionsconsistofflowinletsandexitboundaries,wall,repeating,andpoleboundaries,andinternal faceboundaries
- Interiorsurroundingflowchannel:Itisinsidespaceofenclosurewithoutf1wing,anditisoccupiedwithair.
- Wallsurroundingflowchannel:ItissixfacesoftheenclosureandborderbetweeninteriorsurroundingandF1 wing, acting as the wall.
- Theinletvelocityhasbeensetto133m/s.
- Turbulentintensitysetto5%andTurbulentviscosityratioto 10.

The SolutionMethodstaskpage allows youto specify variousparametersassociated with the solution method to be used in the calculation.

Schemeprovides a drop-down list of the available pressure-velocity coupling schemes: SIMPLE, SIMPLEC, PISO, and Coupled. Fractional Step is available in the drop-down list when the non-iterative time advancement (NITA) scheme is enabled in the Solution Methods task page.

Gradientcontainsadrop-downlistoftheoptionsforsettingthemethodofcomputingthegradient.

- Green-GaussCellBased.
- Green-GaussNode-Based.
- LeastSquaresCellBased.

InthissolvingcaseItuseLeastSquaresCellBased method. LeastSquareCellBasedmethod:

ItcanusetheResidualMonitorsdialogboxtocontroltheresidualinformationthatANSYSFLUENTreports.

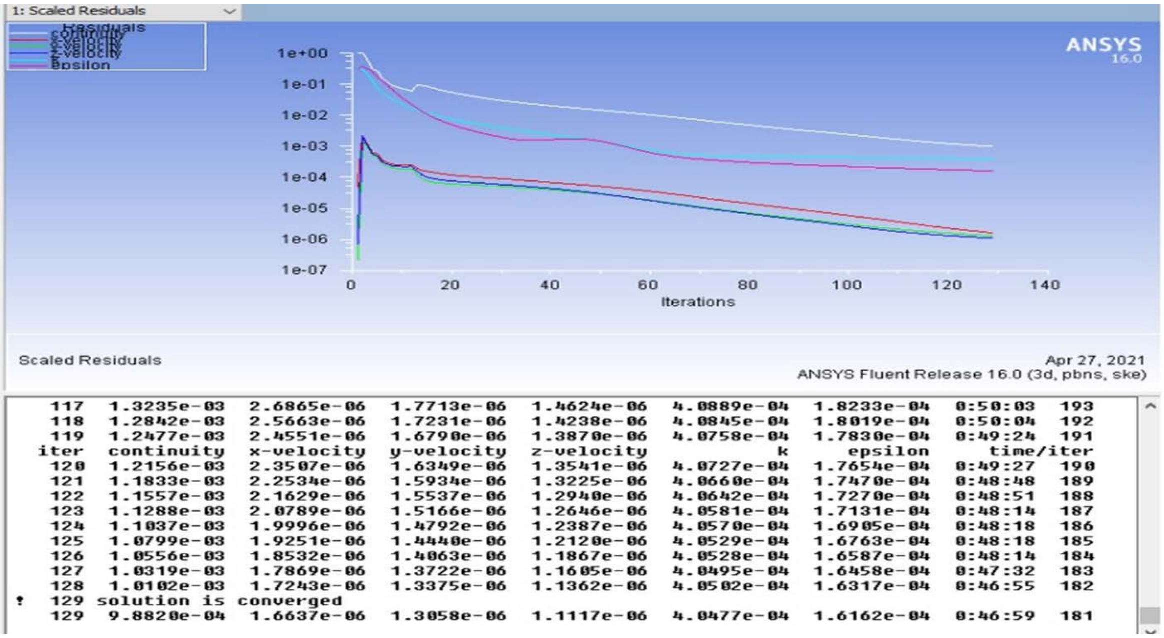
ItcanusetheDragMonitordialogboxtosavetheconvergencehistoryofthedragcoefficientonspecifiedwallzones.

Itcanusethe LiftMonitordialogboxtosavetheconvergencehistoryoftheliftcoefficientonspecified wallzones.Make sure that Plot is enabled in the Options group box.

Compute fromis a drop-down list of zones; the default values for applicable variables will be computed frominformation contained in the zone that you select from this list. The computation will occur when you select the required zone, and the variable values will be displayed in Initial Values. You can also choose the all-zones item in this list to compute average values based on all zones. Reference Frameindicates whether the initial velocities are absolute velocities (Absolute) or velocities relative to the motion of each cell zone (Relative to Cell Zone). This selection is necessary only if your problem involves moving reference frames or sliding meshes. If there is no zone motion, both options are equivalent. Initialize initializes the entire flow field to the values listed. Resetresets the fields to their "saved" values. Select inlet-velocityfrom the Compute From drop-down list.

V. RESULTS AND DISCUSSION

The Run Calculation task page allows you to start the solver iterations. Number of Iterations (for steady flow calculations) sets the number of iterations to be performed. (For unsteady calculations using the explicit unsteady formulation, this will specifythe number of time steps, since each iteration will be a time step.) Start the calculation byrequesting 130 iterations. While the calculation is in progress, a Working dialog boxwill appear. Clicking the Cancel button or typing <Control-C>in the console window will interrupt the calculation (as soon as it is safe to stop). At the end of each solver iteration, the residual sum for each of the conserved variables is computed and stored, thus recording the convergence history. This history is also saved in the data file. The residual sum is defined below. On a computer with infinite precision, these residuals will go to zeroasthe solutionconverges. Onanactualcomputer, the residualsdecayto some smallvalue ("round off") and then stop changing ("level out"). For single-precision computations (the default for workstations and most computers), residuals can drop as manyas six orders of magnitude before hitting round-off. Double-precision residuals can drop up to twelve orders of magnitude. The residuals are the error magnitudes for equations as iterations progress. The equations include the governing equations, I.e. the Navier-Stokes momentumequations for eachdirection(x, y, and z if 3d, or just x and yif 2d), the continuityequation (conservation of mass), and if heat transfer is applicable, the energy equation. The equations may also include equations of the turbulence model defined under models viscous. The residual is the differencebetweenthepreviousresultandthecurrentresult.Astheseerrorsaredecreasingtheequationresultsarereaching values that are changing less and less. This is what is known as convergence. That is the solutions are converging. If these errors begin to increase, the solution is then said to be diverging. Let me give a simplified example of convergence: Initial value = 2, 1st iteration value = 1.5 (residual value is equal to 2 - 1.5 = .5), 2nd iteration value = 1.2 (residual value = 1.5 -1.2 = .3), 3rd iteration value = 1.05 (residual value = 1.2 - 1.05 = .15),... 1000th iteration value = 1.00001, and 1001th iteration value = 1.000005 (residual value = .00001 - .000005 = .000005). Perhaps the solution is converging to a value of 1.Anoppositetrendcouldbegivenasanexampleofdivergence.



Scaled Residuals

Apr 27, 2021
ANSYS Fluent Release 16.0 (3d, pbns, ske)

```
    117  1.3235e-03  2.6865e-06  1.7713e-06  1.4624e-06  4.0889e-04  1.8233e-04  0:50:03  193
    118  1.2842e-03  2.5663e-06  1.7231e-06  1.4238e-06  4.0845e-04  1.8019e-04  0:50:04  192
    119  1.2477e-03  2.4551e-06  1.6790e-06  1.3870e-06  4.0758e-04  1.7830e-04  0:49:24  191
   iter  continuity  x-velocity  y-velocity  z-velocity           k     epsilon     time/iter
    120  1.2156e-03  2.3507e-06  1.6349e-06  1.3541e-06  4.0727e-04  1.7654e-04  0:49:27  190
    121  1.1833e-03  2.2534e-06  1.5934e-06  1.3225e-06  4.0660e-04  1.7470e-04  0:48:48  189
    122  1.1557e-03  2.1629e-06  1.5537e-06  1.2940e-06  4.0642e-04  1.7270e-04  0:48:51  188
    123  1.1288e-03  2.0789e-06  1.5166e-06  1.2646e-06  4.0581e-04  1.7131e-04  0:48:14  187
    124  1.1037e-03  1.9996e-06  1.4792e-06  1.2387e-06  4.0570e-04  1.6905e-04  0:48:18  186
    125  1.0799e-03  1.9251e-06  1.4440e-06  1.2120e-06  4.0529e-04  1.6763e-04  0:48:18  185
    126  1.0556e-03  1.8532e-06  1.4063e-06  1.1867e-06  4.0528e-04  1.6587e-04  0:48:14  184
    127  1.0319e-03  1.7869e-06  1.3722e-06  1.1605e-06  4.0495e-04  1.6458e-04  0:47:32  183
    128  1.0102e-03  1.7243e-06  1.3375e-06  1.1362e-06  4.0502e-04  1.6317e-04  0:46:55  182
!   129 solution is converged
    129  9.8820e-04  1.6637e-06  1.3058e-06  1.1117e-06  4.0477e-04  1.6162e-04  0:46:59  181
```

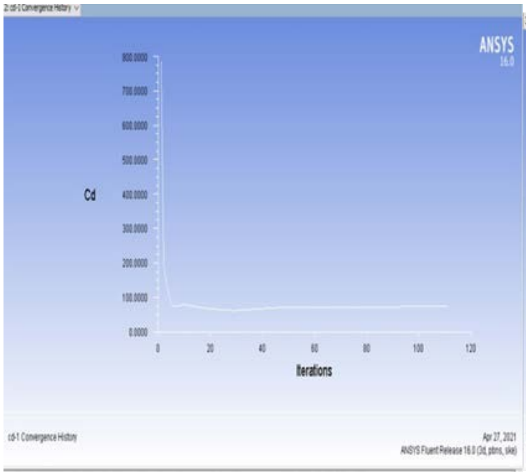
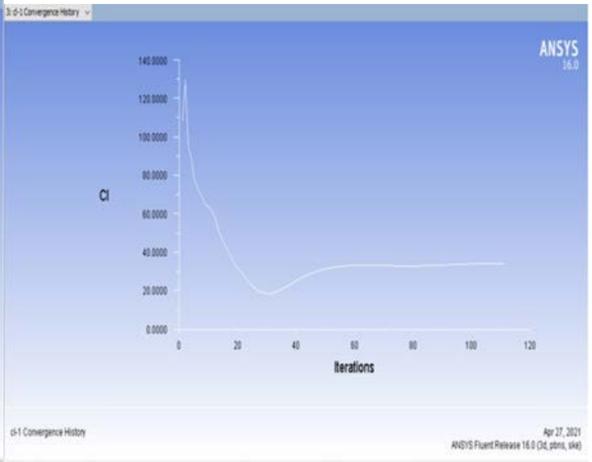


Fig.7.ScaledResidualGraphFig.



8.ConvergencehistoryofDragandLift

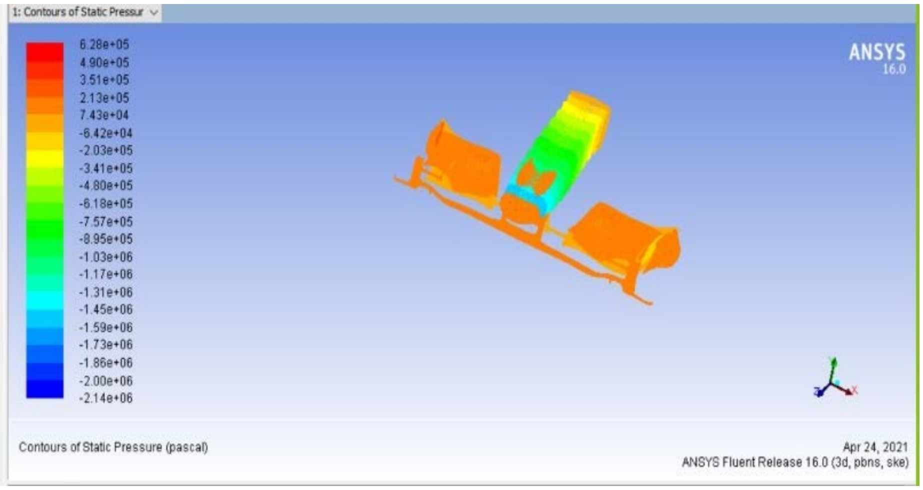


Fig.9ISOviewofcontoursofstaticpressureoninteriorsurroundingflowchannel

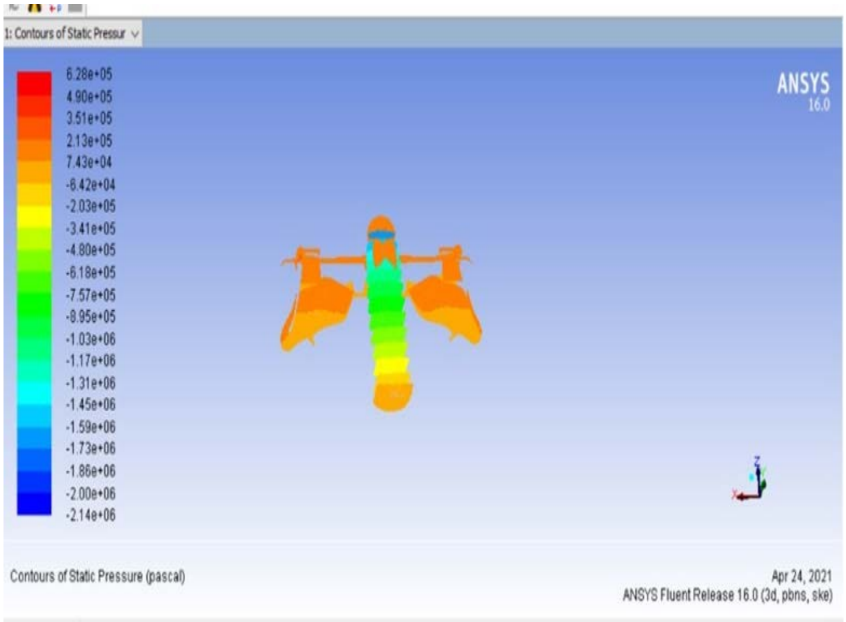


Fig.10Backviewofcontoursofstaticpressureoninteriorsurroundingflowchannel

With ANSYS FLUENT still running, you can perform a simple evaluation of the velocity and temperature contours on the symmetryplane. Later, you will use ANSYS CFD-Post (from within ANSYS Workbench) to perform the same evaluation. Make sure that Node Values is enabled in the Options group box. Select Pressure... and Static Pressure from the Contoursof drop-down lists. Select Interior surrounding flow channel from the Surfaces selection list. Click Display to display the contours in the active graphics window. Select Wall surrounding flow channel from the Surfaces selection list. Click Display to display the contours in the active graphics window.

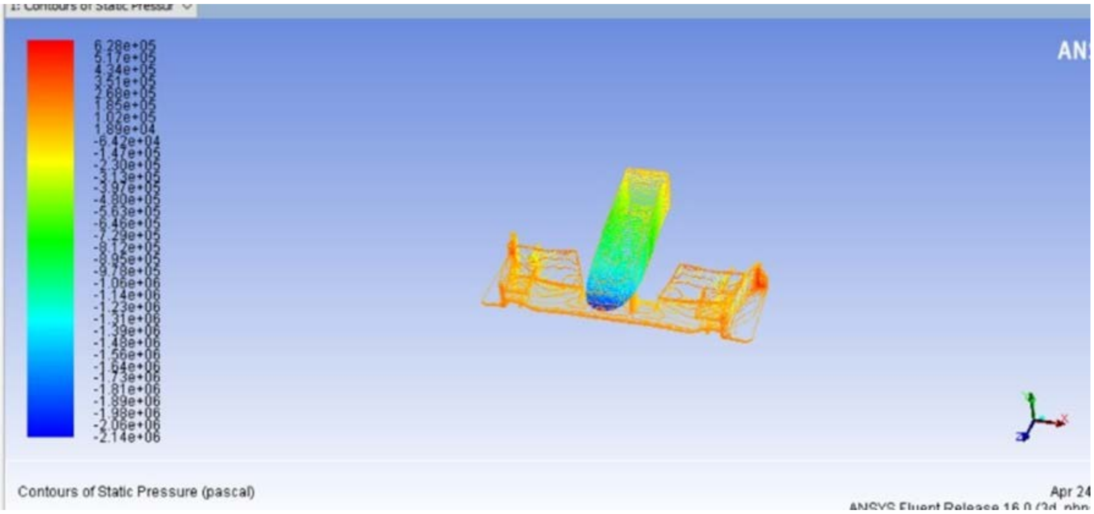
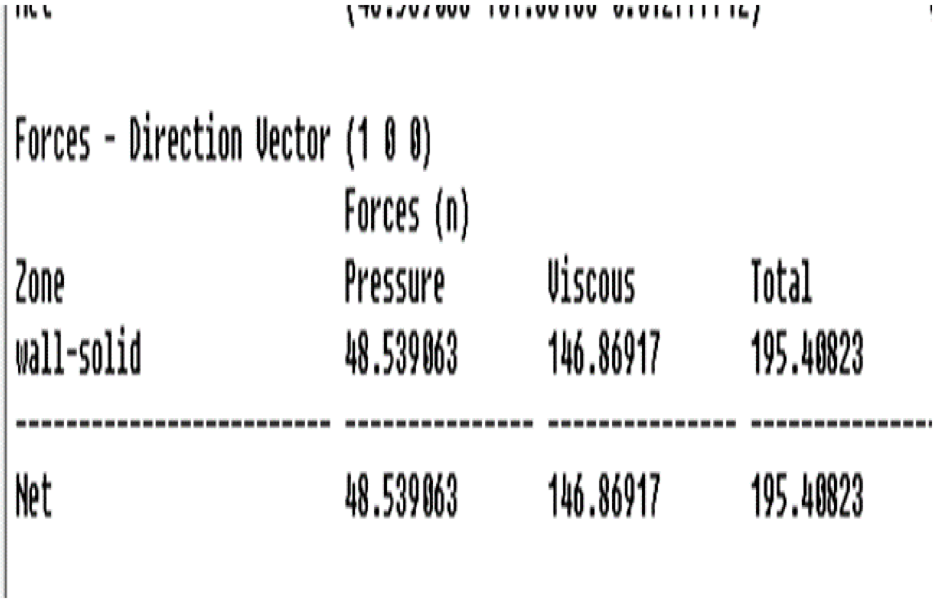


Fig.11. ContoursofStaticPressureonWallSurroundingFlowChannel

Optionscontainradiobuttonsthatcontrolcomputationoftheforces, moments,orcentreofpressure.Forcesenable the computation of the pressure and viscous forces. Moments enables the computation of the pressure and viscousmoments. Centre of Pressure enables the computation of the average location of the pressure. Direction Vector contains the components of the force vector. This label is visible when the Forces radio button is active. X, Y,Zare the components of the force vector along whichthe forces will be computed. Wall Zonescontains a selectable list of wall zones. The force or moment information is printed for each zone, and then a total force or moment for all the zones is presented.

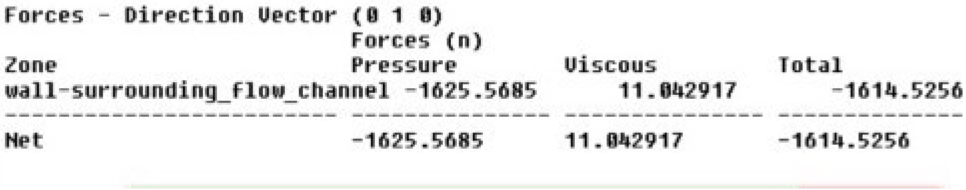
For drag force computation, take direction vectors in x, y, z as 1,0,0 and select write, and it will be displayed on theconsole.



```
Forces - Direction Vector (1 0 0)
                          Forces (n)
Zone                      Pressure       Viscous        Total
wall-solid                48.539063      146.86917      195.40823
------------------------- -------------- -------------- --------------
Net                       48.539063      146.86917      195.40823
```

Fig.11.DragForcevaluesdisplayedonConsolefirstcase

Drag force along xdirectionis 48.53063.For downward force computation, take direction vectors inx, y, z as 0,1,0 and select write, and it will be displayed on the console.



```
Forces - Direction Vector (0 1 0)
                                 Forces (n)
Zone                             Pressure        Viscous         Total
wall-surrounding_flow_channel -1625.5685        11.042917       -1614.5256
------------------------------ --------------- --------------- -------------
Net                              -1625.5685      11.042917       -1614.5256
```

Fig.12.DragForcevaluesdisplayedonConsolesecondcase

STREAMLINESFORFW31FRONTWING

A streamline is a line that is tangential to the instantaneous velocity direction (velocity is a vector, and it has a magnitude and a direction).To visualize thisina flow, we could imagine the motionofa small, marked elementof fluid. For example, we could mark a drop of water with fluorescent dye and illuminate it using a laser so that it fluoresces. If we took a short exposure photograph as the drop moves according to the local velocity field (where the exposure needs to be shortcompared to the time it takes for the velocity to change appreciably), we would see a short streak, with a length V \Delta t, and with a direction tangential to the instantaneous velocity direction. If we mark many drops of water in this way, the streamlines in the flow will become visible. Since the velocity at any point in the flow has a single value (the flow cannotgo in more than one direction at the same time), streamlines cannot cross. except at points where the velocity magnitude is zero, such as at a stagnation point. Streamlines are a very good representation of velocity field.

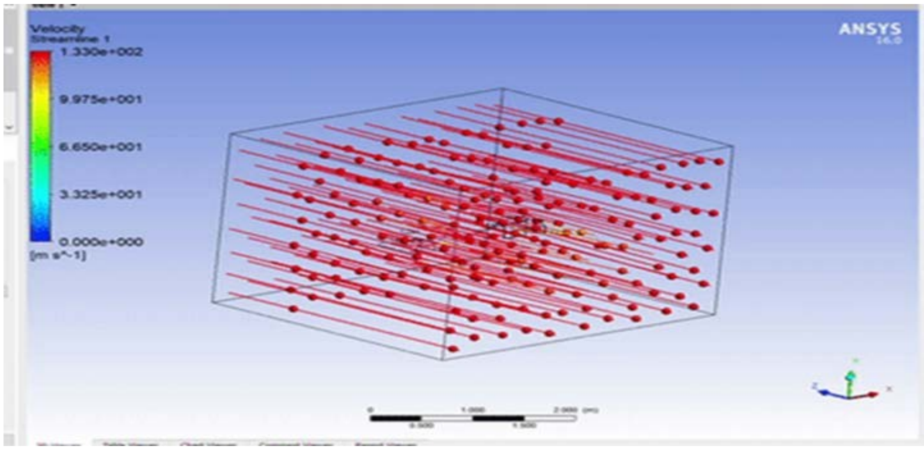


Fig.13.VelocityStreamlineonFrontWing

The aerodynamic lift, drag and flow characteristics of a high-speed William FW 31 front wing were numerically investigated. Where drag force is 48.53063 and downward force (negative lift) is -1625.5685. Due to lack of converged solution, time and CPU consuming for each iteration and lack of having constant CD and CL values; results have showed that the most appropriate turbulence model for external flows around the front wing is $k-\varepsilon\varepsilon$model. We might face some not relevant results if the meshing resolution is not satisfactoryenough. Analysis of the front wing gives a complete picture of key parametersandareas tobefocussedandtheyaretobeunderstoodwith respect tothebettermentofthe performance.The overall pressure over the front wing of F1 is critically analysed and determined using CFD. Where the maximum pressure obtains is 628258.4 Pascal, and the average pressure around the front wing is 76400 Pascal withvelocity of 133 m/s. The F1 wings catch hold the adverse effects of the aero dynamic action and makes an arena for the vehicletodischargeitsfunctionwell. Moreover,theabovestudywashighly educationalregardingthefield’sofaerodynamics,fluiddynamics,CFDsimulationsandautomobiledesign.Furthertheperformanceofthecar anditsaerodynamicswereevaluatedandifthesimulationswerenotuptothemark,thennecessaryrecommendationsandsuggestions were provided. With the aid of Computational Fluid Dynamics (CFD) simulations the development cycle and overall cost were greatly reduced while new and creative ideas were easily tested on the virtual platform with accuracy.

REFERENCES

[1] M. Pavan Kumar, Md. Muzammil, M. Naga Sai, A.Suresh, (2021), Modeling and Computational Fluid Dynamic Analysis of F1 Front Wing using ANSYS, International Journal for Scientific Research & Development, 9(2), 331-335, ISSN (online): 2321-0613, http://ijsrd.com/Article.php?manuscript =IJSRDV9I20162

[2] A Suresh, B Vidhya darshini, CH Srija, J Srujana, & D Arun Sai. (2021). Modeling and CFD analysis of F1 rearwing. International Journal of Advance Research and Innovative Ideas In Education, 7(4), 466-473.

[3] Mohammad Arief Dharmawan, Ubaidillah,brianAqifNuafal,et.al, Feb-2018 “Aerodynamic analysis of formula student car”, AIP conference proceedings,1.5024107.

[4] Shreyas Vaidya,Chinmay Kulkarni, ,Decembar2017, “Aerodynamic development of formula SAE car”,International Journal of Engineering Research & Tevhnology,2278-0181.

[5] Francesco Mariani, Lorenzo scappticci et.al., 2015, “Experimental and Numerical analysis of external aerodynamics”, ELSEVIER, ,1013-1029.

[6] Rizal E.M. Naseer, Firdaus Mohamad et. al, 2012, “Aerodynamics of ARTeC’s PEC Emo-C car”, ELSEVIER, ,1775-1780.

[7] Anirudhha Patil ,Siddharth Kshirsagar, 2014, “Study of front wing of F1 car using CFD”, International Journal of Mevhanical Engineering And Robotics Reasearch, ,2278-0149.

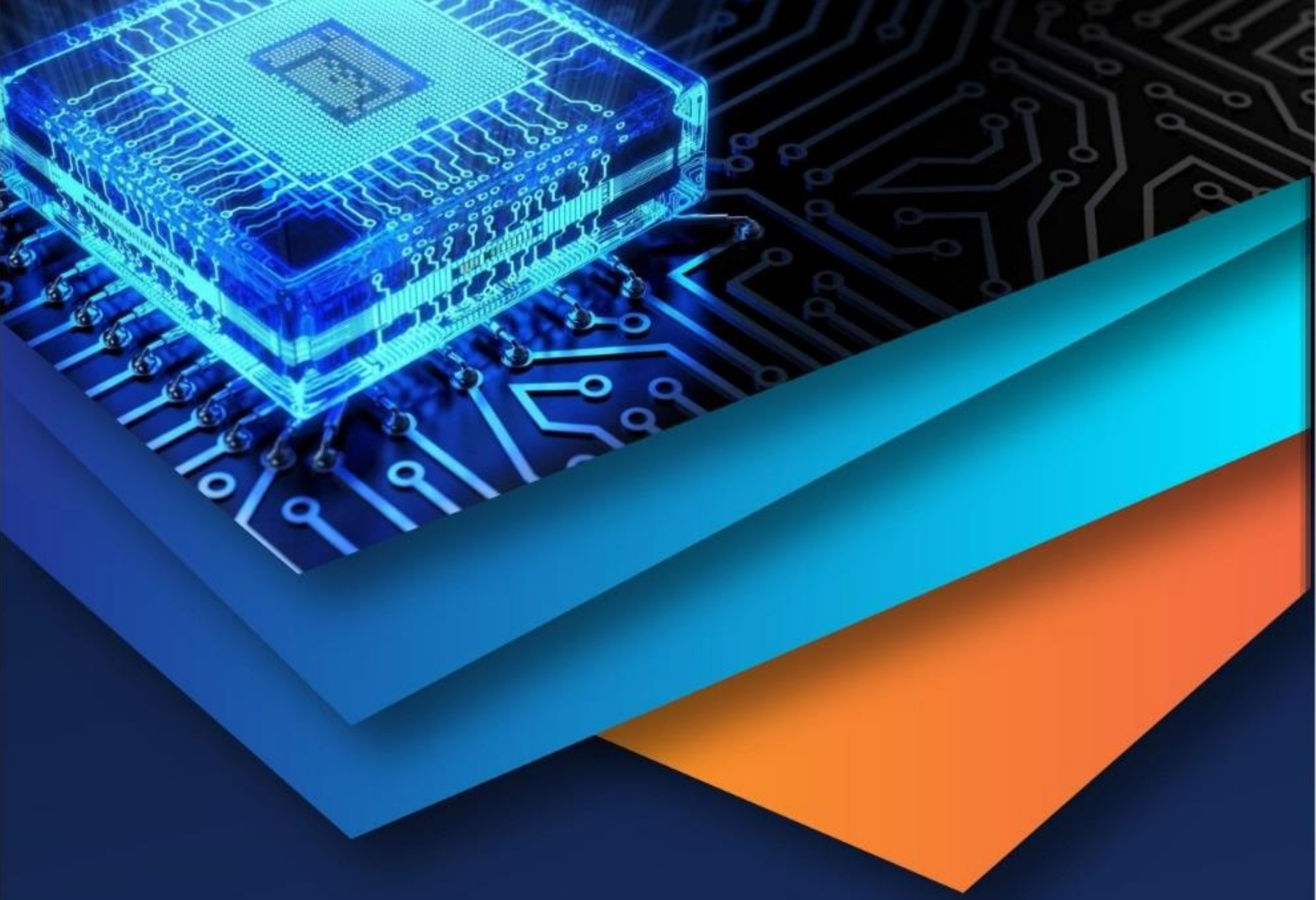
[8] Triya nanalalvadgama, ApritPatelet.al,May-2015,“StructuralanalysisofF1car”, InternationalJournalandResearch in Engineering,Science& Technology, 2394-2444.

[9] S. Pehan and B. Kegl, 2002, “Aerodynamic Aspect of Formula S Racing Car”, International Design Conference-Design, May-2017.

[10] AravindPrasanth, Sadjyot Biswal et.al, July-2016, “Complete Design and Optimization of the Aerodynamics of FSAE Car using Solid works, ANSYS, XFLR5”, World Congress on Engineering, , 2078-0958.

[11] Pulkit Agarwal, Mayur Shrikhande and P. Srinivasan, 2011,'Heat Transfer Simulation by CFD from Fins of an Air Cooled Motorcycle Engine under Varying Climatic Conditions', Proceedings of the World Congress on Engineering 2011 Vol III WCE 2011, July 6 - 8, London, U.K.

[12] MukeshDidwania,GopalKrishan,Ravikant, April2013,'StudyandAnalysisofHeatTransferthroughTwoDifferent Shape Fins using CFD Tool', International Journal of IT, Engineering and Applied Sciences Research (IJIEASR) ISSN: 2319-4413 Volume 2, No. 4.



10.22214/IJRASET

INDEX COPERNICUS
45.98

IMPACT FACTOR:
7.129

TOGETHER WE REACH THE GOAL
IMPACT FACTOR:
7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY