70(10A): 424-429; 2022. DOI: 10.18311/jmmf/2022/32945

Print ISSN : 0022-2755

Journal of Mines, Metals and Fuels

Contents available at: www.informaticsjournals.com/index.php/jmmf

To study the Aerodynamic Effects of Spoiler of a Car (Sedan – Segment) using Computational Fluid Dynamics

Madhu B P[1*] and G Mahendramani[2]

[1]Assistant Professor, School of Mechanical Engineering, REVA University, Bangalore 560064, Karnataka, India.
[2]Associate Professor, Department of Mechanical Engineering, Government Engineering College, Ramanagaram 562159, Karnataka, India.

Abstract

Aerodynamic is an important aspect in consideration with regard to automotive design. Efficient diagnosis of aerodynamic forces' effect on a car body becomes necessary when automobiles run at very high speed. In recent days, almost all automobile products coming into the market are designed considering the effect of aerodynamic forces.

The objective of this analysis is to simulate the flow of air over a car body and to estimate the effect of aerodynamic forces like drag and lift. The prediction of drag becomes necessary for vehicle running at higher speeds, may lead to higher fuel consumption, dropping the efficiency. The induced "LIFT" will disconnect wheels and road, which may lead to accidents. The current analysis we are performing, will analyze the effect of these forces on car, with and without spoiler, to understand the spoiler's presence helping in reducing the drag and its contribution to increase the efficiency of the car. Hypermesh is used for pre-processing and ANSYS Fluent is used as a solver and post processor.

Keywords:Automotive industry, Aerodynamics, ANSYS Fluent, Spoiler of a car, Lift

1.0 Introduction

The automotive industry growing with rapid speed and global competitiveness recommends for the need for improvement in the process of analysis of any engineering problem. With the aid of CAE techniques, the majority of the engineering problems are being solved easily in quick time and optimum effort. The vision of our project is to execute an analysis on the "Spoiler of a car (Sedan - Segment) using Computational Fluid Dynamics to study the aerodynamic effects".

The application of CAE techniques will drastically reduce the time and expenses involved in actual testing of the prototypes. Ultimately, virtual product development plays a significant role in bringing new ideas in a shorter time to market.

Figure 1: Spoiler Installed in a Passenger Car

*Corresponding Author

2.0 Requirement of Analysis

- To simulate the flow of air over a car (sedan segment).
- Estimation of drag and lift forces induced.
- Comparison of results obtained by analysis of car, with and without spoiler.
- To know how spoiler helps in reducing the adverse effect of lift and drag.

Methodology

Figure 2 shows methodology of the CFD anlysis the CAD model is imported into the HYPERMESH graphical interface. The geometry is first checked by the visual inspection and then various features are identified. Now geometry clean up is carried out to ease the process of discretization. Mesh the component according to the given quality parameters.

Once the mesh satisfies quality parameters, deck preparation can be started. The domain can be generated using the information given in the literature survey. After generating the domain of standard dimensions it is exported to *.cas format which is supported by the FLUENT solver. Now it is the time to import the case file into FLUENT 3D solver interface and apply all the boundary conditions on defined locations. After this all the input requirements for the solver (FLUENT) are provided. This file is saved again before starting the iterations.

Once the iterations are carried out solution gets converged, the results can be easily plotted in the post processing stage.

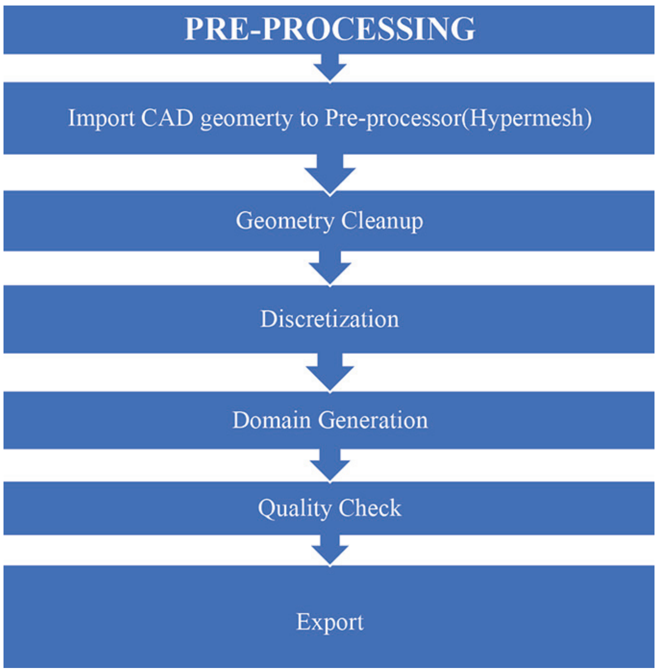


Figure 2: Methodology

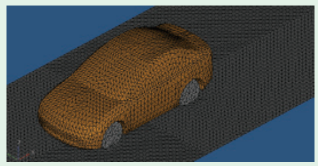
The obtained results can be viewed in the form of graphs and plots using post processor (FLUENT). This provides results with graphical images and solution at any point can be easily determined. Thus the results obtained are compared to respect to the contours of pressure, velocity and turbulence.

Tables 1 and 2 show the geometry and meshed model of car.

Table 1: Input Components

	Input Component Name	CAD Model
1	Car	

Table 2: Meshed Model

	Component Name	Meshed Component
1	Car	

3.0 Calculations

Following calculations are necessary to give appropriate inputs for the solver:

A. Hydraulic Diameter

Hydraulic Diameter (D_H) = $(4 \times A)/P$

Where, A = Cross-sectional area of flow (m^2)

P = Wetted Perimeter (m)

$D_H = (4 \times (5.1 \times 8.5)) / (2 \times (5.1 + 8.5))$

$D_H = 6.375$m

B. Reynolds Number

Reynolds Number (R_e) = $(\rho \times v \times D_H) / \mu$

Where, ñ = Density (kg/m^3), v = Velocity (m/s),

µ = Viscosity (kg/m-s)

For v = 60 kmph:

v = 60 kmph = $(60 \times 10^3) / 3600 = 16.67$m/s

$R_e = (1.225 \times 16.67 \times 6.375) / (1.7894 \times 10^{-05})$

$R_e = 7275191.7542$

For v = 100 kmph:

$v = 100kmph = (100 \times 10^3) / 3600 = 27.78m/s$

$R_e = (1.225 \times 27.78 \times 6.375) / (1.7894 \times 10^{-05})$

$R_e = 12123864.8429$

For v = 140 kmph:

$v = 140kmph = (140 \times 10^3) / 3600 = 38.88m/s$

$R_e = (1.225 \times 38.88 \times 6.375) / (1.7894 \times 10^{-05})$

$R_e = 16968173.6895$

For v = 180 kmph:

$v = 180\ kmph = (180 \times 10^3) / 3600 = 50m/s$

$R_e = (1.225 \times 50 \times 6.375) / (1.7894 \times 10^{-05})$

$R_e = 21821211.0204$

For v = 220 kmph:

$v = 220kmph = (220 \times 10^3) / 3600 = 61.11m/s$

$R_e = (1.225 \times 61.11 \times 6.375) / (1.7894 \times 10^{-05})$

$R_e = 26669884.1091$

4.0 Boundary Conditions

Now it is the time to apply boundary conditions at defined locations in the domain. The Table 3 shows boundary conditions applied.

Table 3: Boundary Conditions

	Particulars	Boundary Conditions
1	Inlet	Velocity Inlet
2	Outlet	Pressure Outlet
3	Top and Side walls	Wall
4	Symmetry	Symmetry
5	Car	Wall
6	Road	Wall

5.0 Results and Discussion

FLUENT after solving gives results in the form of contour and vector plots. FLUENT also acts as a post processor and results in terms of drag and lift can be easily obtained as previously discussed in the post processing part.

Initial analysis was carried out for car without spoiler and for car with spoilers of 3 different profiles with three different angles of attack forming a total of 9 different combinations. For 5 different speeds a total of 50 iterations of analysis are done and their results in terms of drag and lift obtained from the solver are shown in Table 4.

In the above report it can be observed that the results of car with spoiler of NACA2412 airfoil section are optimum at 15° angle of attack compared to its behaviour at other orientations. And results of car with spoiler of NACA0012 airfoil section is optimum at 12.5° angle of attack compared to other two orientations. Similarly the results of car with spoiler of ClarkY airfoil section is optimum at 10° angle of attack.

It is noted that the car with spoiler of ClarkY airfoil section at 10° angle of attack has higher drag force. Since it has a net lift value near to zero it is selected for the further stages of optimization even though it has higher drag.

The results for better understanding are formulated as graphs which are shown in Figs.3 and 4.

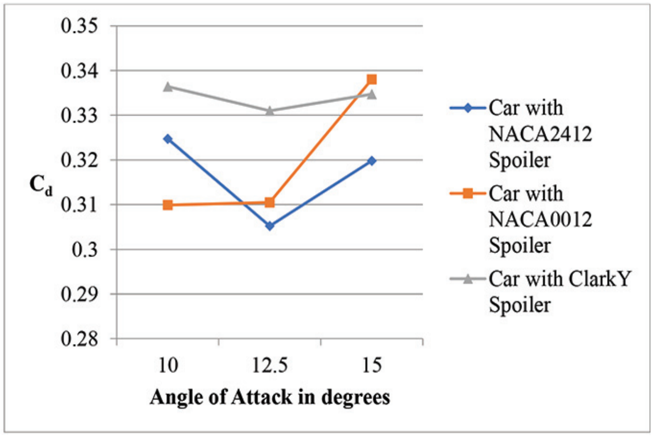


Figure 3: Co-efficient of drag vs Angle of attack

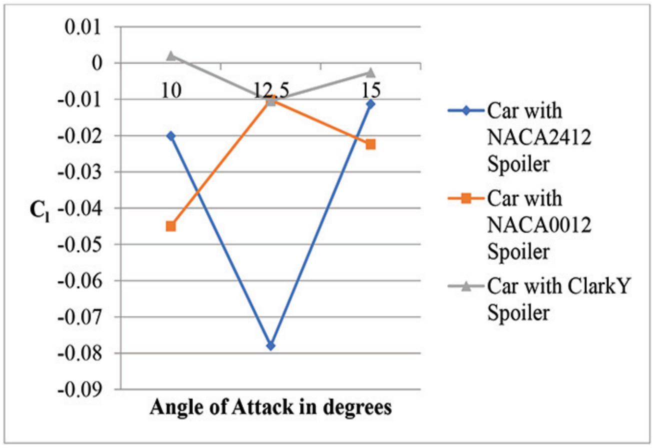


Figure 4: Co-efficient of lift vs Angle of attack

6.0 Results Comparison

A comparison of CFD results in terms of drag and lift for car without spoiler and car with spoiler of ClarkY airfoil section at 10° angle of attack is shown in Table 5.

The contours (Figs.5 to 8) give a better understanding on the difference in results.

Table 4: Aerodynamic effects of spoiler on car

	CAR	Speed	60kmph	100kmph	140kmph	180kmph	220kmph
Without Spoiler		Drag force in N	66.727619	185.0559	361.69368	597.59101	891.2092158
		Co-efficient of drag	0.33768789	0.33718851	0.33649037	0.33613372	0.33560526
		Lift Force in N	9.8259687	26.90216	51.968981	85.160948	126.8039814
		Co-efficient of lift	0.049726196	0.04902796	0.048347712	0.047901433	0.047750946
Spoiler with NACA2412 section	10°	Drag force in N	64.602934	178.90946	349.66927	577.72302	862.4732677
		Co-efficient of drag	0.32693508	0.32600782	0.32530336	0.32495851	0.32478408
		Lift Force in N	-4.1080608	-11.028425	-21.896957	-35.910447	-54.87972369
		Co-efficient of lift	-0.0207896	-0.020095934	-0.020371118	-0.020198962	-0.020666218
	12.5°	Drag force in N	61.98	171.955	336.4427	556.43602	830.55
		Co-efficient of drag	0.314	0.313805	0.31322	0.3102	0.3099
		Lift Force in N	-15.376	-45.985	-93.189	-160.157	-243.26
		Co-efficient of lift	-0.07796	-0.08391	-0.0861	-0.090112	-0.9167
	15°	Drag force in N	62.834185	174.0827	339.84378	564.46505	837.9821813
		Co-efficient of drag	0.31798729	0.31719144	0.31616693	0.3175012	0.3155614
		Lift Force in N	-3.5218726	-10.92015	-15.713612	-20.174082	-53.90858111
		Co-efficient of lift	-0.017823271	-0.021821733	-0.014618848	-0.01134755	-0.020300512
Spoiler with NACA0012 section	10°	Drag force in N	60.644163	167.91	328.54	542.729	809.82
		Co-efficient of drag	0.307	0.306	0.3056	0.3052	0.3049
		Lift Force in N	-9.0568	-27.7816	-57.31	-98.92	-150.97
		Co-efficient of lift	-0.045	-0.05066	-0.0533	-0.0556	-0.0568
	12.5°	Drag force in N	61.5365	170.65	334.187	552.94	824.51
		Co-efficient of drag	0.312	0.311	0.3108	0.31059	0.31048
		Lift Force in N	-2.0142	-5.886	-11.612	-20.15	-30.103
		Co-efficient of lift	-0.01019	-0.0107	-0.0108	-0.01135	-0.0115
	15°	Drag force in N	65.839259	181.94548	356.95298	591.17365	881.7830203
		Co-efficient of drag	0.33319172	0.33153969	0.33207964	0.33252415	0.33205561
		Lift Force in N	-5.5108668	-15.539203	-24.489146	-39.926391	-59.71948748
		Co-efficient of lift	-0.027888758	-0.02831542	-0.022782671	-0.022421163	-0.022488742
Spoiler with Clark Y section	10°	Drag force in N	66.97096	185.15466	361.77862	597.01675	890.66568
		Co-efficient of drag	0.3389195	0.33738598	0.3365706	0.33581117	0.3354005
		Lift Force in N	0.88537158	1.1397096	1.273553	1.9837255	2.9624453
		Co-efficient of lift	0.004480594	0.002076761	0.001184812	0.00111581	0.00111557
	12.5°	Drag force in N	70.141	195.923	339.6651	560.91	950.2955
		Co-efficient of drag	0.35	0.357	0.310600503	0.315	0.3578
		Lift Force in N	-2.50041	-7.347	-11.29	-18.94	-38.005
		Co-efficient of lift	-0.01266	0.013399	-0.0105	-0.0106	-0.0143125
	15°	Drag force in N	66.748144	184.76458	361.01948	596.14587	888.872486
		Co-efficient of drag	0.337792919	0.33667732	0.3358646	0.33532169	0.33473087
		Lift Force in N	-5.1492424	-15.414644	-31.080241	-51.064254	-80.28570325
		Co-efficient of lift	-0.02605876	-0.028088508	-0.028914653	-0.028722755	-0.030233422

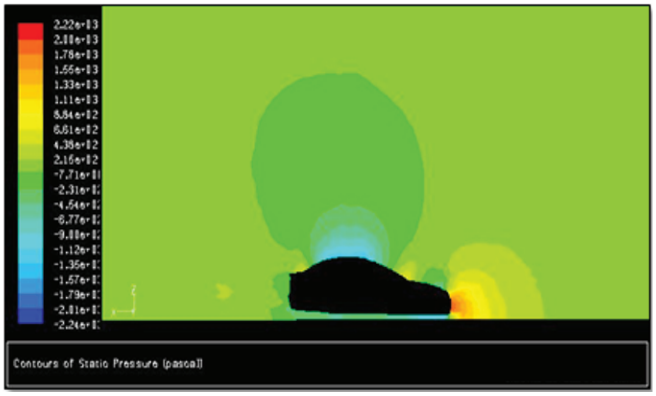


Figure 5: Contours of Static Pressure – Car without spoiler

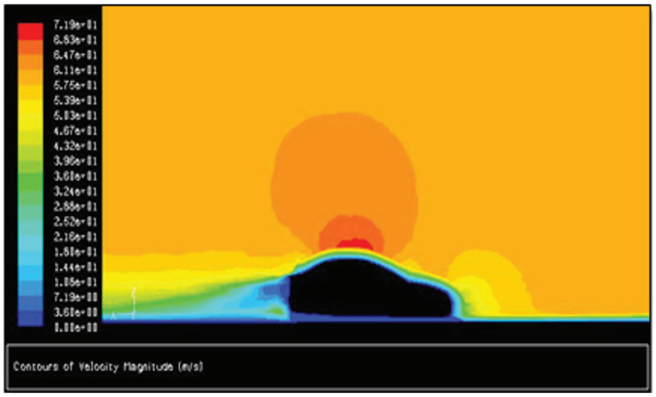


Figure 7: Contours of Velocity Magnitude – Car without spoiler

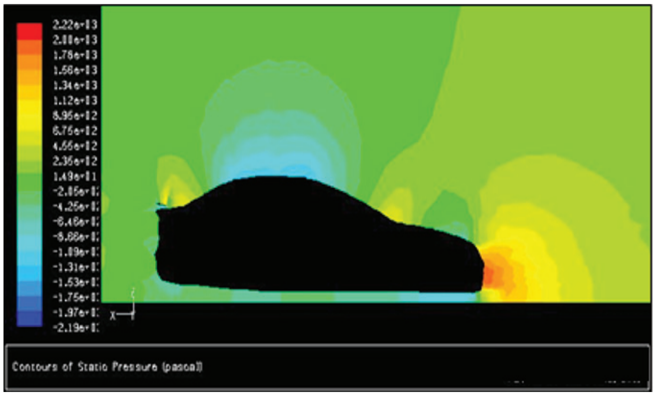


Figure 6: Contours of Static Pressure – Car with spoiler of ClarkY airfoil section at 10^0 angle of attack

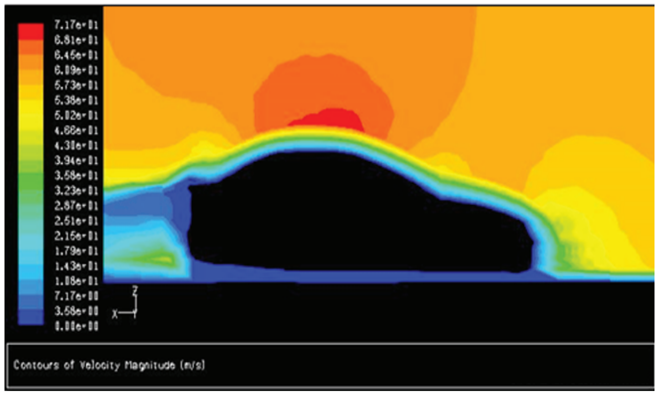


Figure 8: Contours of Velocity Magnitude – Car with spoiler of ClarkY airfoil section at 10^0 angle of attack

Table 5: Comparison of Car without Spoiler and Car with Optimized Spoiler

		Speed	60kmph	100kmph	140kmph	180kmph	220kmph
Car Without Spoiler		Drag force in N	66.727619	185.0559	361.69368	597.59101	891.2092158
		Co-efficient of drag	0.33768789	0.33718851	0.33649037	0.33613372	0.33560526
		Lift Force in N	9.8259687	26.90216	51.968981	85.160948	126.8039814
		Co-efficient of lift	0.049726196	0.04902796	0.048347712	0.047901433	0.047750946
Car with Spoiler of Clark Y section	10°	Drag force in N	63.61552	176.120108	344.24364	568.9184	849.3464875
		Co-efficient of drag	0.3219384	0.32092757	0.32025672	0.3200064	0.31984089
		Lift Force in N	-0.76780507	-1.2332035	-2.884336	-4.0251223	-4.21323411
		Co-efficient of lift	-0.003885623	-0.002247142	-0.002683355	-0.002264059	-0.00158659

The results in graphical representation give a better idea of the concept. A clear comparison in the form of graphs is shown in Figs 9 and 10:

It is very clear from the above report and graphs that a spoiler at the rear gives more stability to the vehicle by reducing its lift and it is a value addition that the final design of spoiler is playing a propitious role in reducing drag force too.

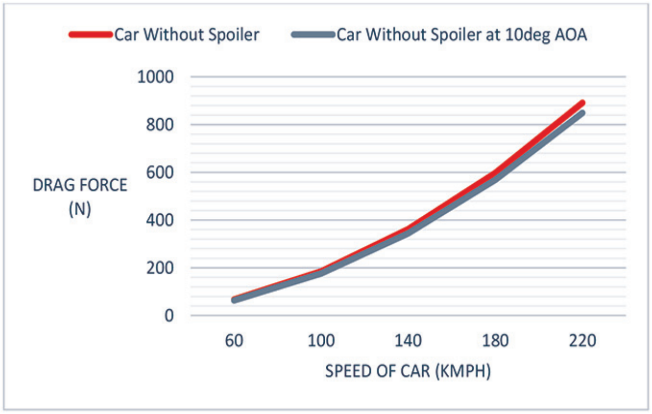


Figure 9: Drag force vs Speed of car

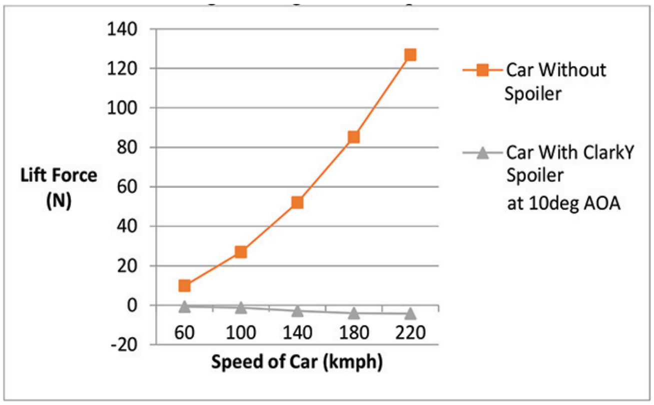


Figure 10: Lift force vs Speed of car

7.0 Conclusions

The objective of this project was to estimate the drag and lift forces induced, comparison of results obtained by analysis of car with and without spoiler and to know how spoiler helps in reducing the adverse effect of lift and drag.

We have run the simulation on car without spoiler and with spoiler at five different speeds i.e., at 60kmph, 100kmph, 140kmph, 180kmph and 220kmph. Here for car with spoiler we have considered three types of spoiler NACA0012, NACA2412 and CLARK Y at three angles of attack i.e., at 10°, 12.5° and 15°.

The results for all the above cases are obtained. By comparing the values of drag and lift forces for both car with and without spoiler, we have seen that lift force is reduced more in case of car with spoiler. Out of the three types of spoiler CLARK Y gives better results.

8.0 References

[1] James, A.E.: Design of an aerodynamic rear spoiler, Federal University of Agriculture, Makurdi, Nigeria (2013).

[2] Redfern, J.: The most aerodynamic new cars you can buy in 2018, Motoring Research (2018). https://www.motoringresearch.com/car-news/most-aerodynamic-new-cars-2018/. Accessed 10 Nov 2019.

[3] Hu, X.-X., Wong, E.T.T.: A numerical study on rear-spoiler of passenger vehicle. *Int. J. Mech. Mechatron*. Eng. 5(2), 636–641 (2011).

[4] Das, R.C., Riyad, M.: CFD analysis of passenger vehicle at various angle of rear end spoiler. Proc. Eng. 194, 160–165 (2017)

[5] Tomar, A.S., Prajapati, A., Sharma, A., Shrivastava, S.: CFD analysis on the aerodynamic effects of spoiler at different angle on car body. *Int. J. Innov. Technol. Explor. Eng*. (IJITEE) 8(7), 2845–2848 (2019).

[6] Ipilakyaa, T.D., Tuleun, L.T., Kekung, M.O.: Computational fluid dynamics modelling of an aerodynamic rear spoiler on cars. Niger. *J. Technol*. (NIJOTECH) 37(4), 975–980 (2018).

[7] Bhatt, P.: Dynamic rear car spoiler for high speed turning stability. *Int. J. Mech. Prod. Eng*. 3(12), 48–49 (2015).

[8] Yuan, Z., Wang, Y.: Effect of underbody structure on aerodynamic drag and optimization. *J. Meas. Eng*. 5(3), 194–204 (2017)

[9] Hamut, H.S., El-Emam, R.S., Aydin, M., Dincer, I.: Effects of rear spoilers on ground vehicle aerodynamic drag. *Int. J. Numer. Meth. Heat Fluid Flow* 24(3), 627–642 (2014.

[10] Stojanovic, N., Abdullah, O.I., Schlattmann, J., Grujic, I., Miloradovic, D.: 3D aerodynamic simulation of the heavy duty commercial vehicle. In: Dumitru, I., Covaciu, D., Racila, L., Rosca, A. (eds.) The 30th SIAR International Congress of Automotive and Transport Engineering, SMAT 2019, pp. 68–81. Springer, Cham (2020). https://doi.org/10.1007/978-3-030-32564-0_9