iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 5 **Issue:** VIII **Month of publication:** August 2017

DOI: http://doi.org/10.22214/ijraset.2017.8014

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor:6.887
Volume 5 Issue VIII, August 2017- Available at www.ijraset.com

CFD Flow Analysis and Optimization of Exhaust Muffler

Saumil Mahesh Trivedi[1], Sangita Bansode[2], Pankaj Pawar[3]
[1]Mtech CAD-CAM & Robotics Student Somaiya college of engineering
[2]Professor, Department of Mechanical Engineering, K.J Somaiya college of engineering,
[3]Manager CAE, HOD FEA, ARK Infosolutions Pvt Ltd.

Abstract:* Silencer is an integral part of the exhaust system. The silencer serves the function of noise and vibration reduction. The exhaust gases in the combustion chamber which are at temperatures of around 1200K are released to the atmosphere at around 323K. Temperature reduction takes place efficiently as the flue gases flow through the exhaust system. In this study, flow analysis is carried out on various geometries and the geometries are checked for the pressure drop and temperature drop based on which the optimum geometry having minimum pressure drop and maximum temperature drop across the flow is selected and considered suitable. The entire flow analysis is done using ANSYS Fluent 18.0. Various Geometry combinations are used considering the minimum pressure drop. These geometries are analysed for flow considering Standard Air, Air as Ideal gas and Real gas as the fluid material for each of the geometries. For all the load cases the geometry which is having minimum pressure drop and maximum temperature drop is considered suitable for structural analysis.
Keywords—Silencer, CFD, Fluent,ANSYS, Flow

I. INTRODUCTION

Silencer is an integral part of the exhaust system. The silencer serves the function of noise and vibration reduction. The exhaust gases in the combustion chamber which are at temperatures of around 1200K are released to the atmosphere at around 323K. Temperature reduction takes place efficiently as the flue gases flow through the exhaust system. Apart from temperature loads, the exhaust system also carries pressure loads, acceleration load and load due to its self-weight. Temperature reduction takes place efficiently as the flue gases flow progressively through the exhaust system. Sound travels in the form of pressure waves which are transverse in nature. So, to reduce the sound, these pressure waves can either be cancelled out or it can be absorbed. There are generally two main reasons for generation of noise in an engine.

A. Noise is generated as the result of the internal combustion
B. Rapid opening and closing of the inlet and exhaust valves generates pressure waves which also generates a lot of noise.

If a vehicle does not have a muffler or silencer than it creates noise due to the difference of frequencies of sound. This noise is undesirable. To reduce this noise arising out of the exhaust from internal combustion engine, mufflers are mandatory device to adopt with the stringent environmental regulations. Muffler contains more pressure hence sound is in the pressure waves.

The primary function of a silencer is to reduce the noise and vibrations and temperature of the exhaust gases flowing through it. Since sound is in the form of pressure waves, controlling the pressure will reduce the noise. So, there was need for a device which could disturb the flow of the exhaust gases flowing through it and also absorb the unwanted noise. As a result, the internal structure of a silencer consists of various baffles and inner pipes having perforations and certain sound absorbing materials so that effective damping and temperature reduction takes place. Based on the function various architectures of silencers are available which are as follows:

1) Absorption type
2) Reflection type
3) Absorption-reflection type
4) Wave cancellation type

ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor:6.887
Volume 5 Issue VIII, August 2017- Available at www.ijraset.com

II. LITERATURE REVIEW

Prof. Amar Pandhare et.al [1] studied the CFD Analysis of Flow through Muffler to Select Optimum Muffler Model for Ci Engine. DragosTutunea et.al [2] studied the CFD analysis of a resistive muffler. Om AriaraGuhan C P et.al [3] presented a CFD Study on Pressure Drop and Uniformity Index of Three Cylinder LCV Exhaust System. R.Ramganesh et.al [4] studied the flow and prediction of back pressure of the silencer using CFD. Ahmed Elsayed et.al [5] carried out the Investigation of baffle configuration effect on the performance of exhaust mufflers. Jianmin Xu et.al [6] carried out the Analysis of Flow Field for Automotive Exhaust System Based on Computational Fluid Dynamics. Claudio Poggiani et.al [7] studied on the Optimization of a fast light-off exhaust system for motorcycle applications.

III. PROBLEM STATEMENT

The main objective of this study is to design an exhaust muffler for a 4 Cylinder petrol engine. The muffler must be designed for the optimum flow of exhaust gases flowing through it. The pressure drop inside the muffler should be minimum whereas the temperature drop through it should be maximum. Necessary specifications required for optimum silencer design for 0.0875 kg/s.

IV. 3D CAD MODEL

Initially design calculations are performed to find out the necessary dimensions of the silencer. Engine capacity and engine speed are taken as a reference for calculations. Based on the dimensions derived, 3D silencer geometry is created.Four geometric configurations are used. The geometry is modified based on the results obtained to meet the pressure drop and temperature drop.

Geometry 1 (Base geometry):

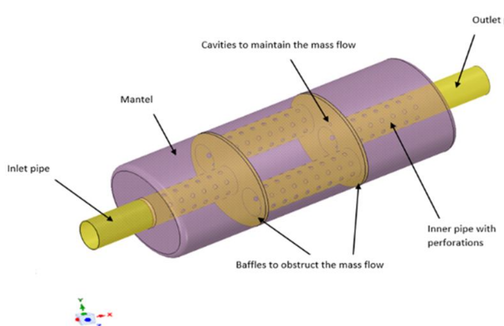


Fig.1 Geometry 1(Base geometry)

Geometry 4 (Optimized geometry from CFD analysis):

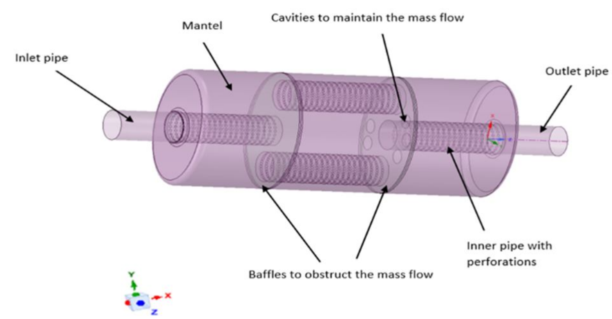
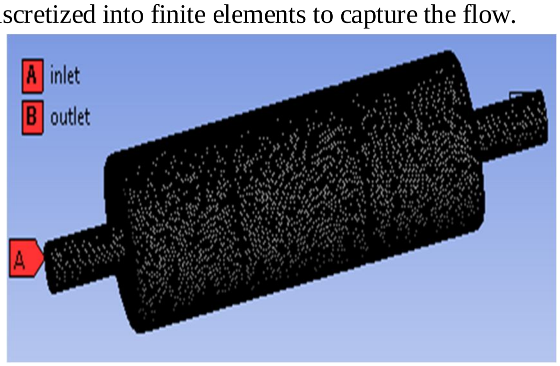


Fig.2 Geometry 4 (Optimized) number of perforations increased

V. MESHING

A. *CFD Fluid volume*

For carrying out the flow analysis on the given geometries, the fluid volume is extracted for each of the geometries and the fluid volumes are discretized into finite elements to capture the flow.



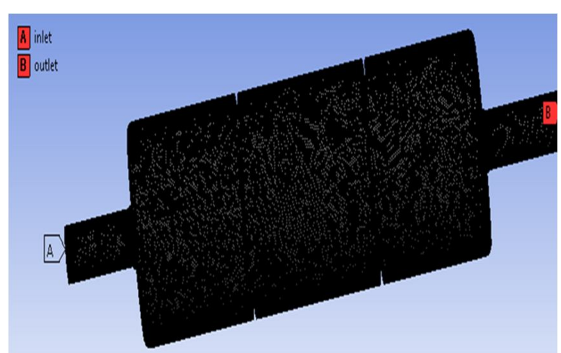


Fig.4 Meshed Fluid Volume Geometry 4(optimized geometry).

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor:6.887
Volume 5 Issue VIII, August 2017- Available at www.ijraset.com

VI. MATERIAL

Material properties of real gas which are obtained from the software are given below.

Table no.1Properties of Real Gas

Real Gas	
Molar mass	28.96 kg/Kmol
Specific heat capacity(C_p)	1006.43 J/kg-K
Specific heat type	Constant Pressure
Reference temperature	25 C
Critical Temperature	132.3 K
Critical Pressure	3758000 Pa
Reference Pressure	1atm
Critical Specific volume	0.002857 m^3/kg
Dynamic Viscosity	1.7894e-05 kg/ms
Thermal conductivity	0.0242 W/m-K

(Ref: from ANSYS FLUENT 18.0 material library)

VII. BOUNDARY CONDITIONS

A. For Fluid Flow

Velocity at the inlet: 50m/s

Pressure at the outlet: 0 Pa (gauge pressure due to atmospheric pressure)

Temperature of the gases at the inlet: 423K

Temperature of free stream air: 298 K

Pressure drop and velocity drop is calculated for all the cases and temperature drop, pressure drop, velocity drop and density distribution is calculated for all the 4 geometries under Real gas condition and the geometry having minimum pressure drop and maximum temperature drop is considered as optimum for fluid flow.

VIII. ANALYSIS AND RESULTS

Flow analysis is carried out on all the geometries using Standard air, air as an ideal gas and real gas as the working fluid.

A. Base case

Flow analysis is carried out on the Geometry 1 using Real Gas as a fluid.

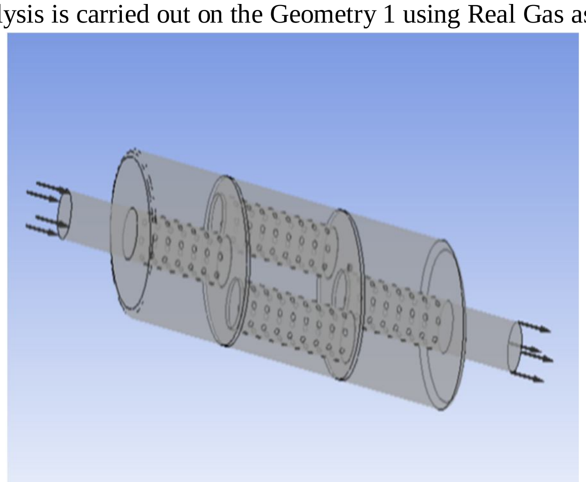


Fig.5 Boundary conditions Case 1 Base Geometry using real gas as a fluid

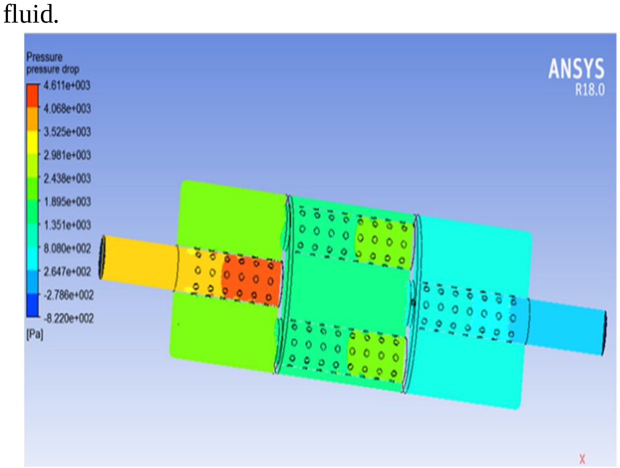


Fig.6 Pressure Contour Case 1 Base Geometry 1 using real gas as a fluid

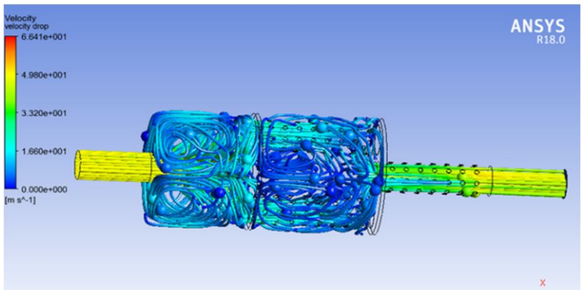


Fig.7 Velocity Streamline Case 1 Base Geometry 1 using real gas as a fluid

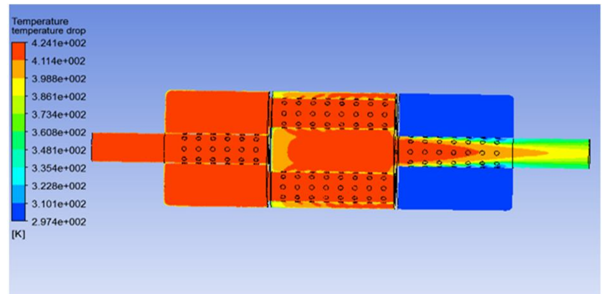


Fig.8 Temperature drop Case 1 Base Geometry 1 using real gas as a fluid

B. Optimized Case

Flow analysis is carried out on the Geometry 4 using Real Gas as a fluid.

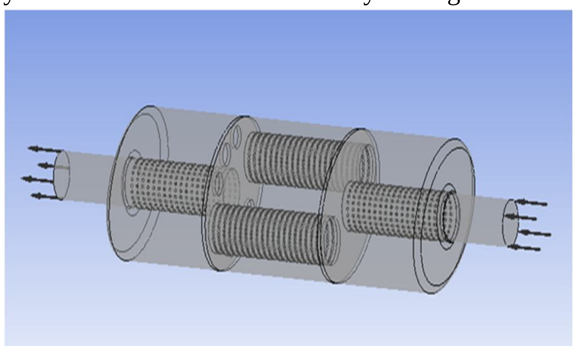


Fig.9 Boundary Conditions Optimized Case OptimizedGeometry using real gas as a fluid

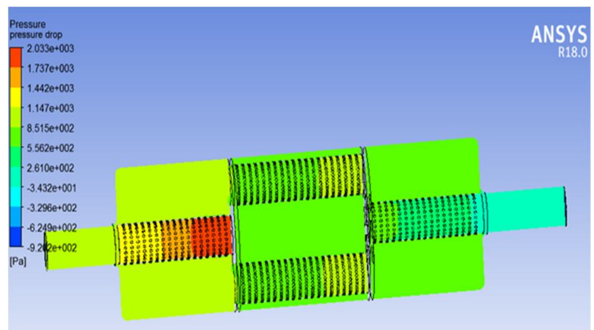


Fig.10 Pressure drop Optimized Case optimized Geometry using real gas as a fluid

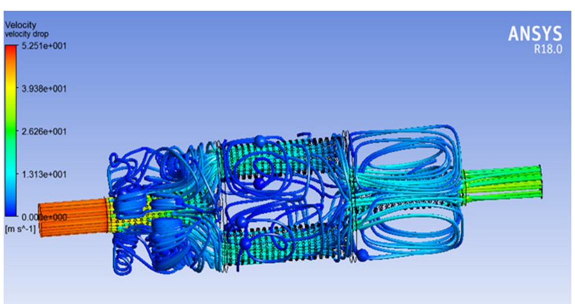


Fig.11 Velocity Streamline Optimized Case Optimized Geometry using real gas as a fluid

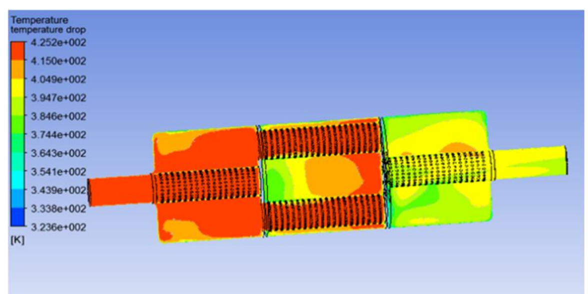


Fig.12 Temperature drop Optimized Case Optimized Geometry using real gas as a fluid

C. Summarized Results

1) *CFD Analysis:* The summarized results of CFD flow analysis carried out on base geometry and final optimized geometry using real gas as the fluid medium are shown in table given below.

Table.2 Summarized Results

Real Gas					
Case	Geometry	Mass Flow	Average Pressure	Average Velocity	Average Temperature
Base case	Geometry 1	Inlet: 0.0875kg/s Outlet:-	Inlet: 3597.06 Pa Outlet: 0 Pa Pressure drop:	Inlet: 49m/s Outlet: 41.6975 m/s	Inlet :423 K Outlet: 366.414 Temperature drop:

		0.0875kg/s	3597.06 Pa		57.702 C
Optimized case	Geometry 4	0.0875kg/s Outlet:- 0.0875kg/s	Inlet: 1072.49 Pa Outlet: 0 Pa Pressure drop: 1072.49 Pa	Inlet: 49m/s Outlet: 27.4497 m/s	Inlet :424 K Outlet: 358.665 K Temperature drop:64.335C

IX. VALIDATION

A. *Temperature at Inlet and Outlet*

1) *Experimental Setup:* The temperatures at the inlet and outlet of the silencer are measured experimentally using a thermocouple measurement in NI LabVIEW.Multiple readings are taken for about 40 seconds and the highest temperature is recorded at the inlet and outlet of the silencer.

Fig no.13 experimental setup for temperature measurement

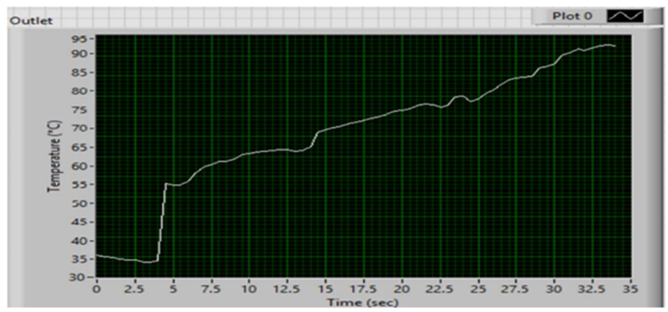


Fig no.14 Temperature readings at the outlet

Once the temperature at the outlet becomes steady, the temperature at the inlet is recorded with the help of non-contact temperature probe.

B. *Result*

The combined results obtained from experimental and simulation.

Table no.3 Combined experimental and simulation results

Experimental results		**Simulation results (obtained from CFD analysis)**	
Temperature at inlet(C)	Temperature at outlet (C)	Temperature at inlet Given as input (C)	Temperature at outlet (C)
149.5	93	150	93.414

C. *Observation*

It has been observed that the experimental and simulation results are in good agreement with each other.

X. CONCLUSION

Flow analysis is carried out on various combinations of geometry using Standard Air, Air as an Ideal gas and Real gas are used as working fluids for fluid volumes for all the geometries. There is no difference in the results obtained using Standard air and Air as an ideal gas as a fluid. Since the working fluid in actual case is a real gas, so flow analysis is carried out using Real gas as a working fluid. Minimum pressure drop of 1072.5 pa and a maximum temperature drop of 64.335 C is obtained for geometry 4 which is validated with experimental results. So, geometry 4 is considered optimum design for the flow.

REFERENCES

[1] Prof. Amar Pandhare, Ayush Lal, Pratik Vanarse, Nikhil Jadhav, Kaushik Yemul, "CFD Analysis of Flow through Muffler to Select Optimum Muffler Model for Ci Engine" , International Journal of Latest Trends in Engineering and Technology (IJLTET), Vol. 4 Issue 1 May 2014, ISSN: 2278-621X.

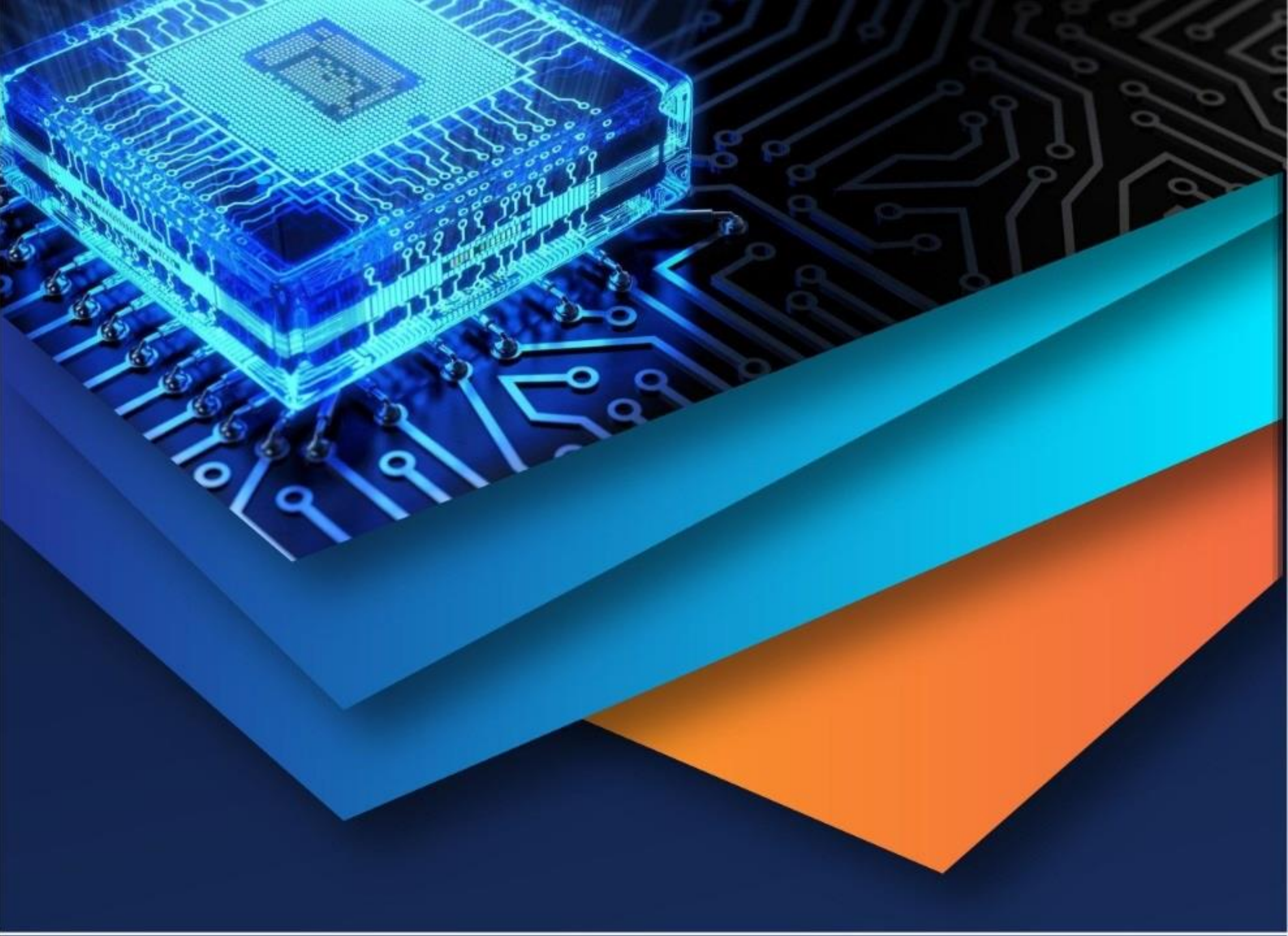
[2] DragosTutunea, Madalina Calbureanu, Lungu Mihai, "Computational fluid dynamics analysis of a resistance muffler", Recent Advances in Fluid Mechanics and Heat & Mass Transfer, ISBN: 978-1-61804-183-8.

[3] Om AriaraGuhan C Pa, Arthanareeswaren G , Varadarajan K N, "CFD Study on Pressure Drop and Uniformity Index of Three Cylinder LCV Exhaust System, International Conference on Computational Heat and Mass Transfer-2015, ScienceDirect Procedia Engineering 127 (2015) 1211 – 1218, doi: 10.1016/j.proeng.2015.11.466.

[4] R.Ramganesh, G.Devaradjane, "Simulation of Flow and Prediction of Back Pressure of The Silencer Using CFD", National Conference On Recent Trends And Developments in Sustainable Green Technologies, Journal of Chemical and Pharmaceutical Sciences www.jchps.com ISSN: 0974-2115.

[5] Ahmed Elsayeda,c , Christophe Bastienb, Steve Jonesa, JesperChristensenb, Humberto Medinac, Hassan Kassemd , "Investigation of baffle configuration effect on the performance of exhaust mufflers", Case Studies in Thermal Engineering, Science Direct, journal homepage: www.elsevier.com/locate/csite, http://dx.doi.org/10.1016/j.csite.2017.03.006.

[6] Jianmin Xu, Shuiting Zhou, "Analysis of Flow Field for Automotive Exhaust System Based on Computational Fluid Dynamics", the Open Mechanical Engineering Journal, 2014, 8, 587-593, 1874-155X/14.



10.22214/IJRASET

INDEX COPERNICUS
45.98

IMPACT FACTOR:
7.129

IMPACT FACTOR:
7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY