CFD Analysis of Flow through Muffler to Select Optimum Muffler Model for Ci Engine

Prof. Amar Pandhare
Head of Department of Mechanical Engineering
Smt. Kashibai Navale College of Engineering, Pune, Maharashtra, India

Ayush Lal
Department of Mechanical Engineering
Smt. Kashibai Navale College of Engineering, Pune, Maharashtra, India

Pratik Vanarse
Department of Mechanical Engineering
Smt. Kashibai Navale College of Engineering, Pune, Maharashtra, India

Nikhil Jadhav
Department of Mechanical Engineering
Smt. Kashibai Navale College of Engineering, Pune, Maharashtra, India

Kaushik Yemul
Department of Mechanical Engineering
Smt. Kashibai Navale College of Engineering, Pune, Maharashtra, India

Abstract - Noise is one of the main contributing factors for environmental pollution. Exhaust noise at the automobile exhaust is the prime contributor towards noise pollution. [1]

Mufflers form an integral part of automobile. Mufflers increase the pressure of the exhaust gases (back pressure) thereby reducing the sound levels of the same. Therefore importance is given to muffler designing and a particular design is selected for which the sound reduction is maximum. For this exhaust gas CFD simulation is carried in softwares such as ANSYS FLUENT.

This paper deals with the designing and flow simulation through the muffler. Two muffler designs have been modeled and CFD gas flow simulation has been carried in both of them. Based upon the gas flow through them, on comparison we have selected the optimum model.

KEYWORDS - Muffler, perforations, shell

I. INTRODUCTION

Using diesel engines mostly as main power element has increased the importance of the technical specification of the diesel engine itself and its other during-and-after design belongings.[2]

The muffler is defined as a device for reducing the amount of noise emitted by a machine.[3] The muffler is engineered as an acoustic soundproofing device designed to reduce the loudness of the sound pressure created by the engine by way of Acoustic quieting. The US Patent for an Exhaust muffler for engines was awarded to Milton O. Reeves and Marshall T. Reeves of Columbus, Indiana of the Reeves Pulley Company on 11 May 1897.

Vehicle population is projected to grow close to 1300 million by the year 2030. [4]
Due to increased environmental concerns requiring less noise emissions combined with reduced emission of harmful gases, it is becoming very crucial to carefully design the exhaust system mufflers for road transport applications. [5]

Exhaust gas emitted from vehicles contains many components that contribute to air pollution, namely carbon monoxide (CO), hydrocarbons (HC) and nitrogen oxides (NO_x).[6]

Muffler consists of four main components

- Inlet pipe
- Outlet pipe
- Shell
- Perforated pipe

When the exhaust gases from inlet pipe pass through the perforations inside the shell, the gases get scattered in different directions. After reflection from the inside surface of the shell, the sound cancellation of waves occurs. The gases pass through the perforations multiple times and even get reflected from the shell surface. Due to the combined effect of these, the level of sound at the muffler outlet is reduced significantly. The flow through the muffler and variation of various parameters such as velocity and pressure along the length of the model can be accurately demonstrated with the help of CFD analysis which display accurate results within a short span of time.

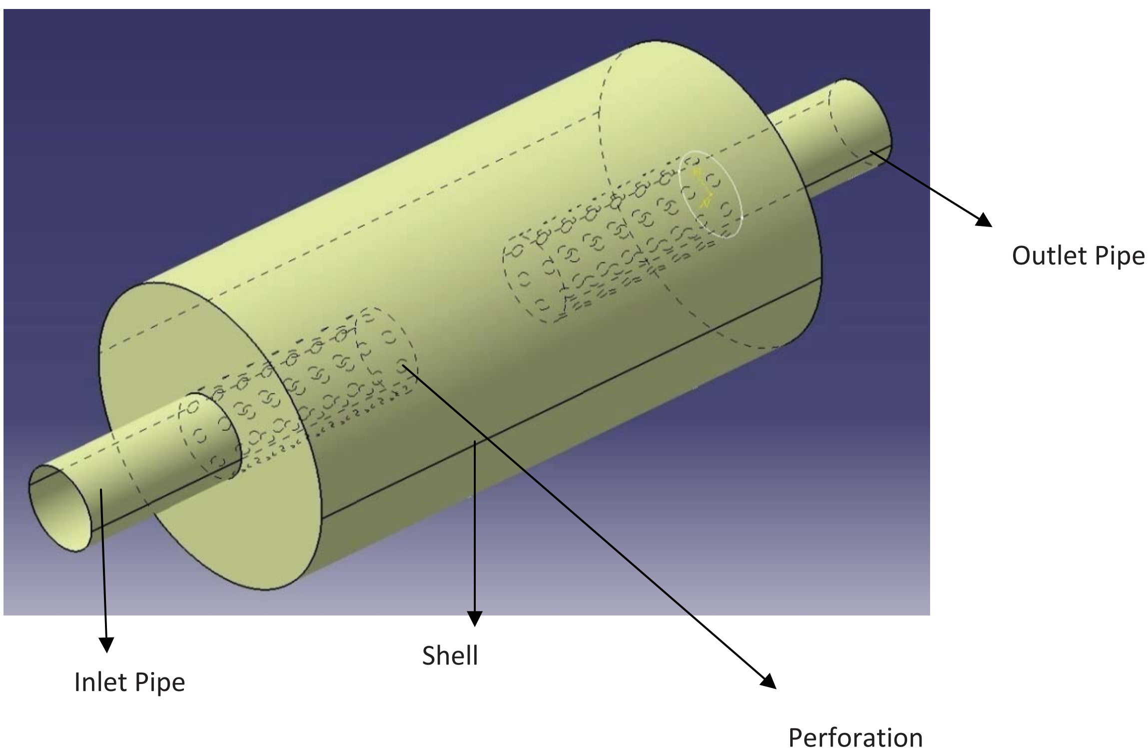

Figure1. Muffler model

II. MODEL GEOMETRY AND MESH GENERATION

The total length of the muffler model is 384mm. The 3D geometry of model prepared in 'CATIA V5 R20' and mesh was generated in 'ICEM CFD'. Number of nodes is 88060. Maximum aspect ratio is 10.4819. There are total 492707 tetrahedral cells.

Model dimensions are as follows:

Diameter of shell=150 mm; Diameter of outlet pipe = 42.1 mm;

Total length (Inlet pipe) = 192 mm; Total length (Outlet pipe) = 192 mm;

Length of inlet pipe (till shell wall) = 85 mm; Length of outlet pipe (Outside shell wall) = 85 mm

We have considered two models for comparison:-

For the 1st model, boundary condition (wall) has been specified at the end of the inlet pipe and also shell, inlet pipe and outlet pipe have been considered as wall.

For the 2nd model, wall has not been specified as boundary condition except for the muffler shell and inlet and outlet pipe.

Pre-processor

The user activities at the pre-processing stage involve:

1) Definition of the geometry of the region of interest: The computational domain.
2) Grid generation-The sub division of the domain in to a number of smaller non overlapping sub domains: a grid (or mesh) of cells.
3) Selection of the physical and chemical phenomena that need to be modeled.
4) Definition of fluid properties.
5) Specification of appropriate boundary conditions at cells which coincide with or touch the domain boundary.

The accuracy of a CFD solution is governed by the number of cells in the grid. In general, the larger the number of cells, the better the solution accuracy.[7]

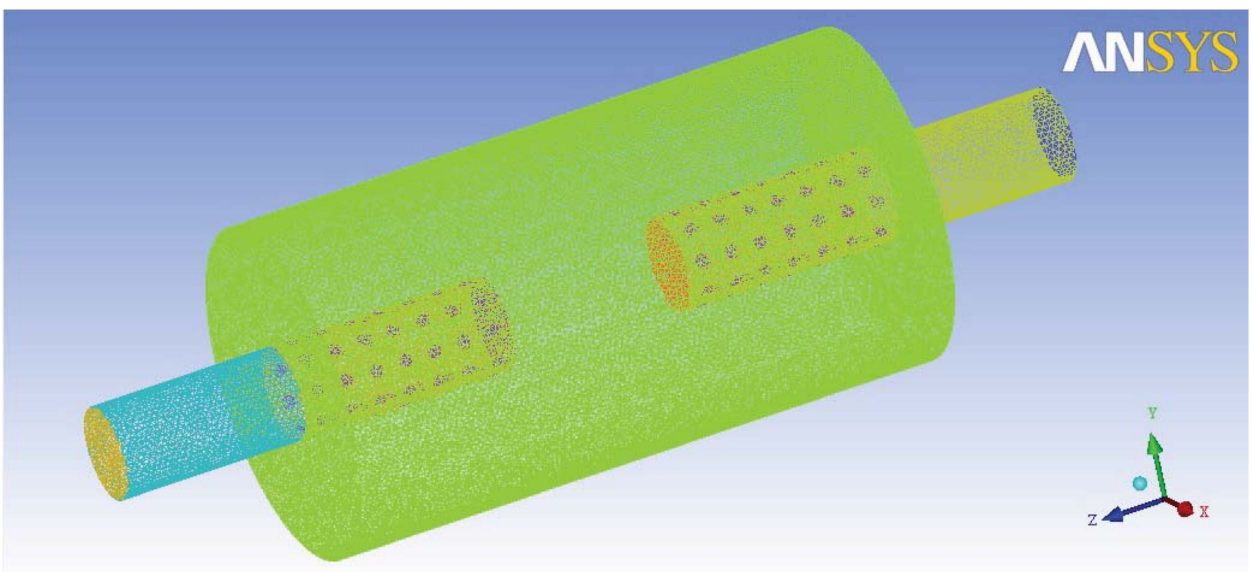

Figure2. Mesh

III. CFD ANALYSIS OF FLOW

Continuity equation (Navier-Stokes) is one of basic conservation equations on which code of Fluent software bases. According to law of conservation of mass in a closed physical system mass of medium cannot neither increase nor decay. Assumption of fluid stream continuity leads to the conclusion that it covers all space of flow (there is so called homogenous flow). On the basis of those assumptions we can make balance mass and in the result we obtain following equation of flow continuity:

$$\partial\rho/\partial t+\nabla .(\rho\ v)=0$$

where:

t -time [s],

v - fluid flow rate [m/s],

ρ -fluid density [kg/m³].

Navier-Stokes equation is used for description of principle of conservation of mass and momentum of flowing fluid. [8]

The Realizable k-e model is used and the energy equation is enabled for both models.

The SIMPLEC algorithm is used for coupling pressure and velocity fields for the 1st model while COUPLED algorithm is used for the 2nd model. As muffler is developed for diesel engine, hence for fluid flowing through the muffler air has been selected [9] the properties of which have been considered at temperature of 500K. The mass flow rate of air at inlet is 10.555e-3 kg/s. The hydraulic diameter at inlet is 0.0421m.

For reference values following data has been considered for air in both the models which has been computed from inlet

Area (m^2)	0.00139
Density (kg/m^3)	0.696
Enthalpy (J/kg)	503400
Length (mm)	384
Pressure (Pa)	300000
Temperature (K)	500
Velocity(m/s)	10.9748
Viscosity (kg/ms)	2.7e-05
Ratio of specific heats	1.4

The solution settings for 1st model are as follows:

Scheme	SIMPLEC
Solution initialization	From inlet
Number of iteration specified	1000

The solution settings for 2nd model are as follows:

Scheme	COUPLED
Solution initialization	From inlet
Number of iteration specified	40000

IV. RESULTS AND DISCUSSION

For model 1

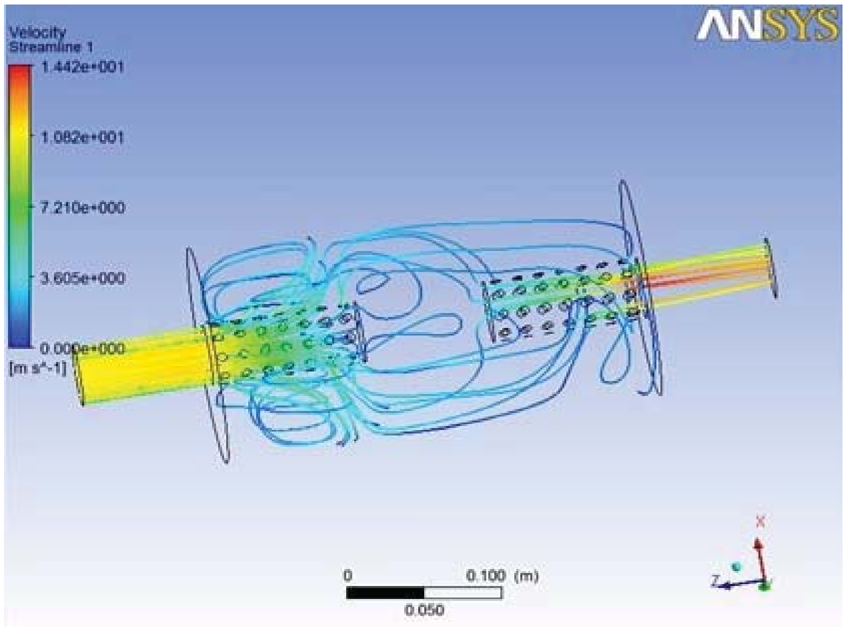

Figure3. Velocity streamlines

Total Pressure is the pressure at the thermodynamic state that would exist if the fluid were brought to zero velocity and zero potential. In the postprocessing, the total pressure is presented as gauge pressure, for compressible and incompressible flows.

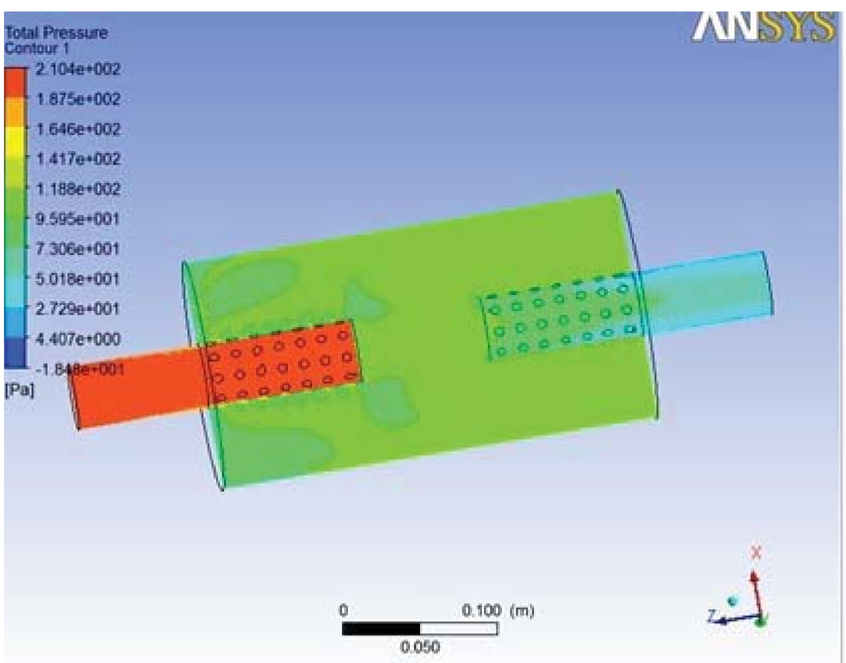

Figure4. Total pressure contour

As we can see in Fig2, as wall has been provided at the end of inlet pipe, the pressure is maximum till the end of inlet pipe.

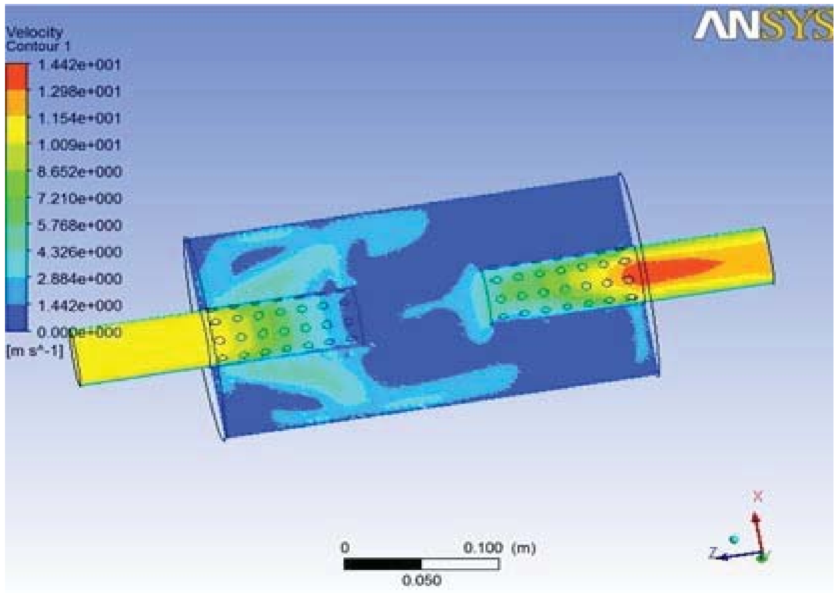

Figure5. Velocity contour

For model 2

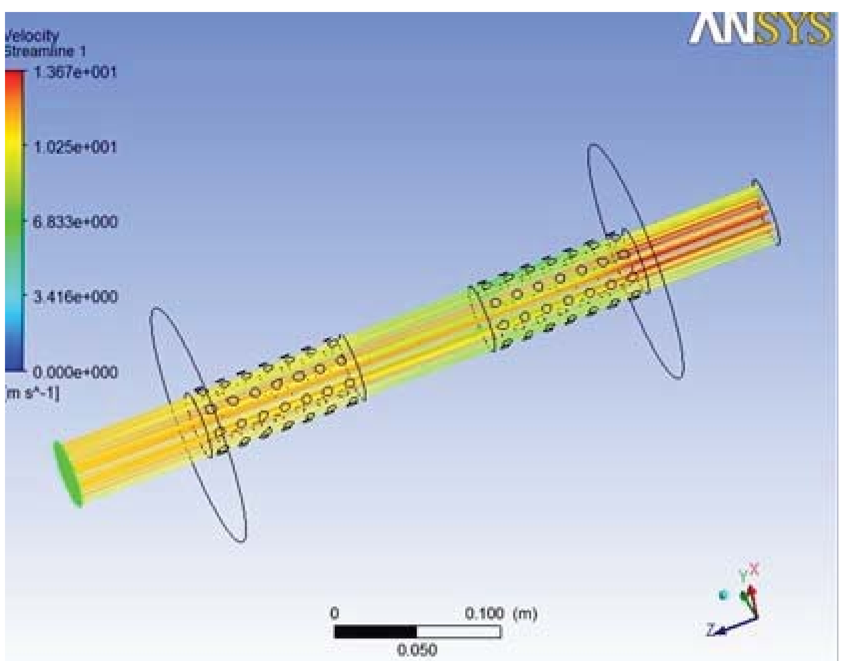

Figure6. Streamline

As we can see there is very less flow through the perforations if wall is not provided at the end of the inlet pipe. The flow through the perforations is essential for sound reduction of the exhaust gases.

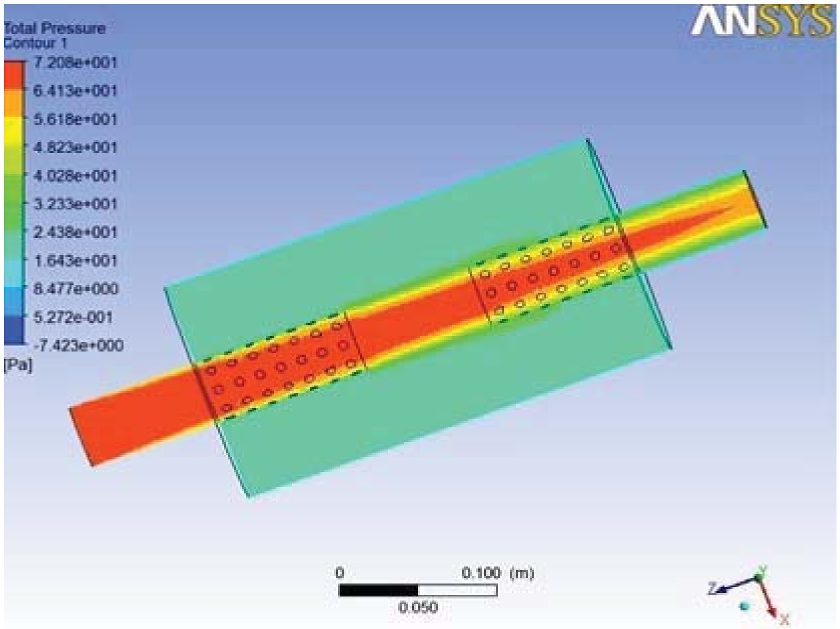

Figure7. Total pressure contour

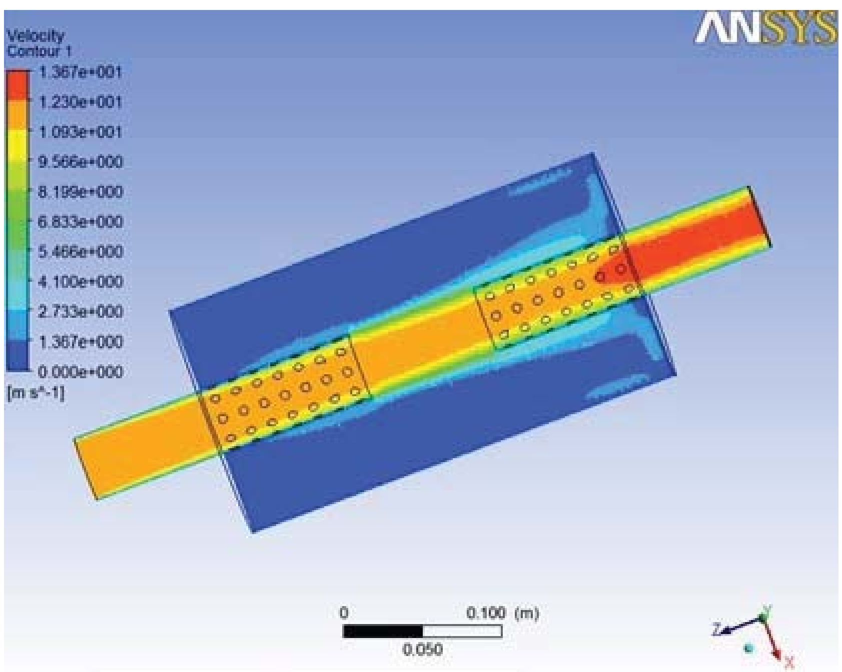

Figure8. Velocity contour

V. CONCLUSIONS

Various conclusions drawn from the observations are as follows:

1. Model 1 of muffler is more efficient than model 2 for noise reduction as can be seen by comparing figure1 and figure2. Streamlines for model1 (figure3) are passing through the perforations which is not the case with model 2 (figure6). Therefore model1 is efficient than model 2 as flow of gases through perforations is necessary as has been explained in muffler working in introduction.
2. For model1, pressure is maximum till the end of inlet pipe (figure4) while in model 2 (figure7), pressure is maximum in the central portion of the muffler and there is negligible flow through the shell. Hence this clearly demonstrates that model 1 is more efficient for noise reduction than model 2 as model 1 offers more reduction in gas pressure and hence reduces noise levels.

REFERENCES

[1] Shital Shah, Saisankaranarayana K, Kalyankumar S. Hatti, Prof. D.G. Thombare, "A Practical Approach towards Muffler Design, Development and Prototype Validation", SAE Paper 2010

[2] Mehmet Avcu, Şadi Kopuz, Mehmet Teke, "MTU 16V 4000 M90 BRAND/MODEL DIESEL ENGINE EXHAUST SYSTEM DESIGN", Journal of Naval Science and Engineering, 2010, Vol. 6, No.1, pp. 39-58

[3] M.Rajasekhar Reddy, Dr K.Madhava Reddy, "Design And Optimization Of Exhaust Muffler In Automobiles", International Journal Of Engineering Research and Applications (IJERA) ISSN: 2248-9622 www.ijera.com Vol. 2, Issue 5, September- October 2012, pp.395-398

[4] Prof. Bharat S. Patel, Mr Kuldeep D. Patel, "A Review paper on Catalytic Converter for Automotive Exhaust Emission", International Journal of Applied Engineering Research, ISSN 0973-4562 Vol.7 No.11 (2012)

[5] Sara Elsaadany, "Investigation and optimization of the acoustic performance of exhaust systems"

[6] H. Santos, M. Costa, "Evaluation of the conversion efficiency of ceramic and metallic three way catalytic converters", Energy Conversion and Management 49 (2008), 291–300

[7] Amar P. Pandhare, Dr. Atul S. Padalkar, Experimental And Simulation Study Of Diesel Engine For Lower Exhaust Emissions, SAE Paper 2013

[8] Wojciech Chuchnowski, Jaroslaw Tokarczyk, Krzysztof Stankiewicz, Mariusz Woszczynski, "Method for modeling temperature distribution in exhaust system of diesel engine in the light of mine systems of heat recuperation", Journal of KONES Powertrain and Transport, Vol. 18, No. 2 2011

[9] Hannu Jääskeläinen, Diesel Exhaust Gas, DieselNet Technology Guide