PAPER • OPEN ACCESS

Numerical analysis of exhaust gas braking system to maximize the back pressure

To cite this article: S Balasubramaniam and D Senthil Kumar 2019 *IOP Conf. Ser.: Mater. Sci. Eng.* **577** 012133

View the article online for updates and enhancements.

You may also like

- Flow characterisation in an exhaust manifold of a single-cylinder internal combustion engine (ICE)
 N A Aziz, M T A Rahman, N A M Amin et al.
- <u>The methodologies and instruments of</u> <u>vehicle particulate emission measurement</u> for current and future legislative <u>regulations</u> Yoshinori Otsuki, Hiroshi Nakamura, Masataka Arai et al.
- <u>Design of Diesel Oxidation Catalyst</u> <u>Temperature Control System Based on</u> <u>Fuzzy Adaptive PID</u> Ning Wang, Yao Sun, Yunfeng Hu et al.





DISCOVER how sustainability intersects with electrochemistry & solid state science research



This content was downloaded from IP address 18.116.118.198 on 05/05/2024 at 05:49

IOP Publishing

Numerical analysis of exhaust gas braking system to maximize the back pressure

S Balasubramaniam^{1*}and D Senthil Kumar^{2*}

¹M.Tech in Engineering Design, Department of Mechanical Engineering, Amrita school of Engineering, Coimbatore, Amrita Vishwa Vidyapeetham, India. ²Asst.Professor, Department of Mechanical Engineering, Amrita school of Engineering, Coimbatore, Amrita Vishwa Vidyapeetham, India.

*E-mail: d_senthilkumar@cb.amrita.edu,subramaniambala453@gmail.com

Abstract. The excessive load on foundation brakes are shared by exhaust brakes during downhill performance of trucks. The exhaust brake increases the life of foundation brakes and also provides assistance to the foundation brake, especially during downhill operation. In this work, the position of the exhaust flow flap have been optimized accordingly to increase the back pressure generated in the diesel engines manufactured at Tata Cummins Ltd., India. The exhaust valve lift with respect to crank angle is modified for exhaust brake module to facilitate back pressure inside engine cylinder. A 1-D GT-POWER six cylinder engine model is modelled to replicate conditions at the inlet of the exhaust circuit. CFD simulations are carried out for various exhaust flow flap positions in order to maximize the back pressure required at engine cylinder. The position of the exhaust flow flap is crucial in resulting back pressure as the exhaust elbow geometry have sharp bends and diverging cross section. The exhaust flow flap is always placed normal to the plane of exhaust elbow. Numerical simulations are performed for various position from the turbo fan. High absolute pressure is observed, when exhaust flow flap is moved closer to the turbo fan. The high absolute pressure at exhaust manifold inlet is observed at 27mm from the turbo fan.

Keywords. Exhaust brake, 1D- 3D simulation, CFD analysis, Regenerative braking

1. Introduction

Significant growth of electric vehicles and extinction of fossil fuels lead to alternate braking system. The engine and exhaust brakes can be considered as potential replacement for conventional braking used in vehicles. The engine and exhaust brakes uses power absorbing mechanism. The alternate braking system also faces challenge in achieving brake power generated compared to the conventional brakes. Many factors affect the performance of the alternate braking system such as engine speed, exhaust brake position and design.

1.1. Literature review

Engine brakes and exhaust brakes cannot completely stop the vehicle but helps in vehicle retardation, which is crucial during downhill levels. They are mostly used in light and medium sized trucks. The exhaust brakes also help in preventing foundation brakes from overheating, which ends in brake failure in many cases. The exhaust brake consists of a butterfly valve which is actuated by Electronic control

Content from this work may be used under the terms of the Creative Commons Attribution 3.0 licence. Any further distribution of this work must maintain attribution to the author(s) and the title of the work, journal citation and DOI. Published under licence by IOP Publishing Ltd 1

unit (ECU) of the vehicle. Vivek Srivastava et al [1] modelled exhaust brake operating by electronic control unit (ECU) and compared with experimental results. The higher back pressure is generated in low engine speeds and does not varies above 1500 rpm [1]. Celso argachoy et al [2] modelled 1-D engine models to study the effects of engine brake. The brake performance is studied by simulating flow flap actuation and valve train [2]. The exhaust brake also reduces the amount of emissions in exhaust gas as inlet valve is closed during the exhaust flow flap actuation. The average retarding pressure generated depends on several factors like engine displacement volume, engine speed, engine friction power and retarding pressure over engine piston [3]. The factors except retarding pressure are fixed over specific time. The retarding pressure over the cylinder piston depends on exhaust manifold geometry and back pressure generated due to presence of exhaust brake. The importance of back pressure in exhaust brake performance makes the position of exhaust flow flap in exhaust elbow more significant. Narayana Reddy et al [4] developed a 1-D thermodynamic model simulation to analyse the braking performance. The presence of turbofan in domain requires specific methods for numerical simulation. The turbofan can be modelled by two methods: moving reference frame method (MRF) and sliding mesh model. In moving reference frame method, mesh does not move. The flow equations are solved with respect to moving reference frame. Computational power needed for MRF method is quite low. Galindo et al [5] performed frozen rotor approach of MRF method in ANSYS FLUENT.

In this work, exhaust flow flap position is optimized to increase the brake efficiency. The numerical simulation is performed at various exhaust flow flap position using ANSYS FLUENT package. The CFD domain is concentrated to the exhaust manifold, turbo charger and exhaust elbow to reduce computational cost and time. The inlet conditions for the CFD domain is generated through 1D six engine cylinder model is developed using GT-POWER package. The results from the 1D six cylinder engine model is fed to the CFD domain consisting of exhaust manifold, turbo charger and exhaust elbow. The exhaust flow flap distance from the turbofan is varied and numerical analysis is performed to optimize the flow flap position.

2. Numerical Formulation

The 1-D model development and CFD domain configurations with applied boundary conditions are discussed in the sub-sections.

2.1. Geometrical configuration

The computational domain contains exhaust manifold, turbocharger with turbo fan, exhaust elbow and exhaust flow flap as shown in figure 1. The exhaust manifold collects the pressurized exhaust gas from the engine to the turbocharger. It collects the exhaust gases from each cylinders and delivers to the respective collector. The turbocharger have compressor and turbo parts. The compressor part intakes air from the ambient domain. The rotating compressor part results in turbo fan rotation increasing the exhaust gas flow velocity. The velocity of the flow results in boosting the pressure at the inlet of engine cylinders, which increases overall efficiency of the engine. The exhaust elbow connects the turbo charger with exhaust pipe. The varying cross section and elbow bend affects the generation of back pressure at exhaust manifold inlet. The flow flap restricts the exhaust flow and generates back pressure in the domain. The flow flap is moved normal in exhaust elbow plane.

2.2. 1-D Model development

The inlet conditions for the CFD domain is generated through 1D six engine cylinder model development using GT-POWER package. GT-POWER simulation software package solves one dimensional equations of flow. GT-POWER is widely used in engine performance test with many commercial engine models. The exhaust gas at the inlet of the exhaust manifold is transient and the mass flow rate varies among the six exhaust manifold inlets due to engine lag. The GT-POWER six cylinder model is modelled to generate boundary conditions provided at exhaust manifold inlet. One way coupling between GT-POWER six cylinder model and 3D exhaust CFD model is assumed. Species transportation from 3D exhaust CFD model to GT-POWER six cylinder model is neglected. The GT-POWER six cylinder model is neglected.

POWER six cylinder model consists of inlet manifold system, six cylinder engine and exhaust port connecting exhaust manifold of 3D exhaust CFD model to exhaust valve of 1-D model.

2.2.1. Inlet manifold system. Env-1 object provides ambient conditions like pressure, temperature to the circuit. The ambient air flows through cooler-1, which is a Piperound object of 300mm length, diameter of 2.6mm. The Piperound object is used to represent pipes with round cross-section in GT-POWER.



Figure 1. Computational Domain

Discretization length of 40 mm is assigned to the cooler-1 considering explicit solution method, where 0.4 times the cylinder bore diameter is considered for intake system. The cooler-1 is calibrated for desired temperature and pressure drop. The cooler-fsplit-1 is a flowsplit object in GT-POWER. The flowsplit object is used to connect single pipes with multiple pipes. The cooler-fsplit-2 is a flowsplit object similar to cooler-fsplit-1 connected to the intercooler outlet. Bellmouth-1 and bellmouth-2 are orifice connections used for smooth transition between intercooler.

The inlet manifold is modelled by discretizing the model into flowsplits and piperectangle objects. The piperectangle object is used to represent pipe with rectangular cross-section. The length of the flowsplit objects is decided by explicit solution method. The intake-split-1 to intake-split-6, which are flow split objects have length of 40 mm is modelled from explicit solution method. The flowsplit objects cannot be further discretized. Each flowsplit objects is considered as a sub-volume. The remaining length of the intake manifold is divided into respective piperectangle objects (Manifoldpipe-1 to Manifoldpipe-5). Intakerunner-01 to Intakerunner-06 are series of piperound objects are connecting intake manifold pipes to respective engine cylinders are shown in figure 2. Experimental data of valve lift for subsequent crank angle is fed to each of the intake valves is shown in figure 3 (a).

2.2.2. Six cylinder engine model. In figure 2, cylinder1 to cylinder 6 represent the six engine cylinders of the model. Initial state, wall temperatures, heat transfer model are defined for each cylinder of the model. *Di-inject-1* to *Di-inject-6* are the injection connection to respective engine cylinders. The injection connection comprises inputs of injected mass per cycle, injected fluid temperature, injection timing with respect to top dead center firing in terms of crank angle and injection duration.

The engine cylinders are controlled by EngineCrankTrain object, which comprises of engine specifications like engine type, firing intervals between the cylinders in terms of crank angle and cylinder geometry is described in Table-1.



Figure 2. GT-POWER six cylinder engine model

exhaustvalve-01 to exhaustvalve-06 are shown in figure 2, which controls the flow of the exhaust from the cylinder by considering the sequence of valve lift, which is obtained experimentally. Exhaust valve is kept open during the compression stroke are shown in figure 3 (c), which improves the performance of the exhaust braking system. Exhaust valve lift with respect to crank angle during absence of exhaust brake is shown in figure 3 (b). The fuel injection is turned off when exhaust brake is applied, which reduces the power generated during power cycle. The initialization of the model is user-imposed. The maximum simulation duration for the six cylinder model is 10 cycles. The simulation shuts-off once steady state is achieved. Solver type for the model is explicit Runga kutta method.

2.2.3. User defined function. The user defined function is a C programming language function that can be hooked to a specific ANSYS FLUENT solver to perform certain features. The user defined function can be interpreted or compiled. Polynomials are generated from the average velocity for varying time plots for each exhaust ports and patched over the faces of respective inlets through user defined function. The user defined function for each inlets will have different user defined function name and respective polynomial obtained.

Initial engine pressure	2.4 bar
Initial temperature	300 K
Cylinder temperature	500 K
Ignition delay	3 crank degrees
Engine bore	100 mm
Engine stroke	100 mm
Connecting rod length	220 mm
Compression ratio	16.5
Engine type	Four stroke
Firing intervals	120 crank degrees





Figure 3. (a) Intake valve lift varying with crank angle; (b) Exhaust valve lift varying with crank angle without exhaust brake; (c) Exhaust valve lift varying with crank angle with exhaust brake

2.3. Boundary conditions

CFD analysis is carried out to evaluate back pressure in the domain. The CFD simulations are performed using ANSYS FLUENT'15 software package. The temperature of the exhaust gas for the whole domain is considered as 600 K from diesel engine data. The objective is to study the variation of back pressure generated by varying the position of flow flap which is situated at the exhaust elbow, so energy equations are neglected and fluid is considered as incompressible. Constant density of 0.58 kg/m³ is considered with respect to the exhaust gas temperature.

2.3.1. Moving reference frame method. The moving reference frame method is considered for the modelling of turbo fan cell zone in the turbocharger section. The impact of turbo fan rotation is limited to certain cell zones near the turbo fan so absolute velocity formulation is recommended. The boundary zone of the turbo fan is modelled as a rotating wall relative to the cell zone, which rotates at a frame motion of 13090 rad/s from diesel engine data. The maximum angular velocity at which turbofan can rotate is considered for analysis. The governing equations for the absolute velocity formulation are:

Conservation of mass:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot \rho \vec{v}_r = 0 \tag{1}$$



Figure 4. Grid independence analysis

Conservation of momentum:

$$\frac{\partial(\rho v)}{\partial t} + \nabla \cdot (\rho \vec{v_r v}) + \rho [\omega \times (\vec{v} - \vec{v_t})] = -\nabla p + \nabla \cdot \vec{\tau} + \vec{F}$$
(2)

where $\vec{v_r}$ is the relative velocity (velocity with respect to moving reference frame), \vec{v} is the absolute velocity (velocity with respect to stationary frame), p is the static pressure, $\vec{\tau}$ is the stress tensor, \vec{F} is the external body forces, $\vec{v_r}$ is the translational velocity.

2.4. Mesh domain

Tetrahedral meshes are used to solve computational domain. The smaller cells theoretically produce better results. Cell refinement is carried out, which increases the cell aspect ratio. The domain is meshed with tetrahedral cells and by octree mesh method. The position of the flow flap is varied and meshed accordingly. The various positions of exhaust flow flap is meshed, which is shown in figure 5. The turbocharger part have finer meshes to solve the equations.

2.5. Code validation

Galindo et al [6] performed 1D-3D simulations in order to reduce the computational cost. The CFD simulations are limited to the required turbocharger domain. Galindo et al [5] modelled turbofan by moving reference frame method with a constant rotating speed. Su et al [7] performed turbomachinery simulations using RANS turbulence model for an incompressible case. The RANS model is selected for affordable computational cost. Shaila sarda [8] performed numerical simulation of exhaust brake without considering turbocharger for RANS turbulence model with tetrahedral mesh.

2.6. Grid Independence

Grid independence analysis is conducted to avoid the discretization error, which decreases in cell size increase. The pressure at inlet of the exhaust manifold at coordinates (-0.0940648, 0.379931, -0.114626) is compared for coarser and finer meshes. Pressure value at specified coordinates does not change significantly, when mesh elements is increased from 5.2 million elements to 6.5 million elements.

2.7. Definition of parameters

SIMPLE-C (SIMPLE- Consistent), which is a pressure based segregated algorithm is used in this model. The pressure based segregated algorithm solves the governing equations sequentially. SIMPLE-C

ICONAMMA2018

IOP Conf. Series: Materials Science and Engineering 577 (2019) 012133 doi:10.1088/1757-899X/577/1/012133

improves the convergence rate in complicated geometries. The discretization equations require face values of the cell, so different interpolation schemes are used to obtain face values. Second order pressure interpolation scheme is applied. Second order upwind discretization scheme is considered for convection-diffusion equations. The absolute convergence criteria of order 1e-5 is used. The solution is initialized using hybrid initialization method, which initializes the variables using various boundary interpolation method and averaged values. The number of time steps for the simulation is assigned as 166 and time step size is 0.001 seconds. The time taken by the engine to complete one cycle is 0.033 seconds.



Figure 5. Mesh domain at various exhaust flow flap positions

3. Results and discussions

The average velocity for each inlet of exhaust manifold are shown in figure 6(a)-6(e), which is obtained from 1-D GT-POWER simulations. The average velocity profile is fed to input of 3D CFD model through user defined function.

The CFD simulations are carried out for an interval distance of 20 mm throughout exhaust elbow profile. Steady state simulations for low rotational speed are performed before the transient state simulations to stabilize the FLUENT calculations. Large complex forces in the flow occurs due to the high rotational speed of the turbo fan region frame. Transient simulations with high rotational speed are performed after solution convergence of the steady state simulations. The interval distance is reduced to half the value, when peak back pressure is observed. Peak Back pressure of 135215.3 Pa is observed at distance of 27 mm from the turbo fan. The high back pressure at the exhaust manifold inlet implies the same amount at the piston. The high pressure at the piston head increases the brake power at the respective cylinder, which reduces the engine speed resulting in vehicle retardation. The simulation illustrates a trend of increase in absolute pressure at exhaust manifold inlet when the valve approaches closer to turbo fan

and attains a peak value as shown in figure 7. After attaining a peak value, the absolute pressure starts to decrease as the exhaust brake approaches the turbo fan. The pressure drop across the turbo fan varies with respect to mass flow across the turbo fan. As the flow flap move away from the turbo fan, vorticity



Figure 6. Average velocity for varying time at various exhaust manifold inlets

is generated in the exhaust elbow due to change in cross section and elbow bend. The elbow causes change in adjacent velocity layers leading to vorticity evolution. The viscosity along the wall diffuses



Figure 7. Absolute pressure at exhaust manifold inlet for varying distance from turbo fan

the vorticity at normal axis of the exhaust elbow. The vorticity increases the flow velocity vector along the elbow resulting in increase of mass flow rate across turbo fan.

4. Conclusions

The optimum position of the exhaust flow flap is to be identified to maximize back pressure at the exhaust manifold inlet is the main objective of the work.1-D GT-POWER cylinder model is modelled to replicate the inlet conditions at exhaust manifold inlet. CFD simulations are performed for various exhaust flow flap positions with the obtained velocity output from the 1-D GT-POWER cylinder model. Maximum absolute pressure at exhaust manifold inlet is observed, when the exhaust flow flap is placed 27mm from the turbo fan.

- Vorticity generation is observed , when exhaust flow flap is placed after the bend in exhaust elbow
- Decrease in back pressure is observed, when flow flap is moved away from turbo fan.

In future various parameters like exhaust flow flap design, valve lift can be studied to increase the braking efficiency.

References

- [1] Vivek Srivastava, Shankar Patil and Mahesh P 2013 SAE No. 2013-26-0125.
- [2] Frederico Augusto Alem Barbieri, Ederson Claudio Andreatta, Celso Argachoy and Hildebrando Brandao 2010 *SAE* No. 2010-01-1531.
- [3] Akiba, Kishiro, and Masahiko Ohtani 1981 SAE No. 810344, 1981, doi:10.4271/810344.
- [4] Narayana Reddy J, Ramkumar D V, Yogesh G Bolar and Ravi T S 2013 SAE No.2013-01-2844.
- [5] Galindo J, Hoyas S, Fajardo P & Navarro R 2013 Engineering Applications of Computational Fluid Mechanics, 7:4, 441-460, DOI: 10.1080/19942060.2013.11015484.
- [6] Galindo J, Tiseira A, Fajardo P, Navarro R 2011 Mathematical and Computer Modelling 54(7-8):1738-1746.
- [7] Su WT, Li FC, Li XB, Wei XZ, Zhao Y 2012 Engineering Applications of Computational Fluid *Mechanics* 6(3):356-365.
- [8] Shaila sarda, Bindu R S 2014 International Journal of Mechanical And Production Engineering, ISSN: 2320-2092.
- [9] ANSYS (2011). ANSYS FLUENT Theory Guide. ANSYS Inc.
- [10] Vishnu P, Thirumalini S., and Ajaykrishna, R 2012 International Journal of Electrical Engineering and Technology (IJEET) vol. 3 no. 1 pp. 316-325.
- [11] Thirumalini S, Lakshmikanthan, C and Dhandapani S 2009 *International Journal of Applied Engineering Research* vol. 4 no. 7 pp. 1369-1384.