TRANSIENT THERMAL SIMULATION PROCESS OVER A DIESEL EXHAUST SYSTEM DURING REGENERATION

Juan Carlos Martínez Laurent[1*], Jorge Alejandro Estevez[1], José Carlos Miranda[2]

[1] General Motors de México, Ingeniería, HVAC/PTC/Thermal, Av. Industria Minera #700, Col. Zona Industrial Toluca.
[2] ITESM-Toluca, Eduardo Monroy Cárdenas, Col. San Antonio Buenavista.

[*]Phone (722) 2765102, juancarlos.martinezlaurent@gm.com

ABSTRACT

With the constant need to meet new environmental regulations, the improvement of automotive exhaust systems technologies to be cleaner and more effective is a necessity.

To achieve these regulations the automakers have been focused on the development of better particle filters and more effective cleaning processes.

Catalyst Oxidation and Diesel Particle Filter Regeneration are good and effective techniques to accomplish these objectives but the amount of heat generated by these processes is a concern in the thermal management of the vehicle.

To gain a better understanding of these effects Finite Element Thermal Analysis has proved to be a useful tool to predict and observe the increment of temperature during these processes.

This work is focus in a simulation process using several 1-D and 3-D techniques to predict the skin exhaust temperature during the regeneration process moment in which the system achieve the maximum temperature.

The objective of this work assesses the simulation results and compares it to physical test results in order to correlate the proposed methodology.

1.-Introduction

Computer Assisted Engineering (CAE) tools have become more important for the automotive industry to predict and improve their products in the early stages of design due the computer increasing power in the last years. This has allowed a significant reduction of testing reducing cost and optimizing the development stages and design process.

Finite Elements methods and Finite Volume methods like Computational Fluid Dynamics (CFD) are CAE tools that have grown in popularity in the automotive industry, but as any CAE tool, CFD must be correlated to physical models or historical test data to have adequate boundary conditions and model validations in early stages of the development process.

In the early steps of the development for an automotive program when the product is brand new, all this information is not available. Making prototypes could be prohibitory in terms of cost, and since calibrations for the engines are still in the very early stages, the information is not confident enough to make a good correlated prediction with the final product, so making assessments could be very risky and inaccurate.

In order to solve this issue in the early development process new procedures must be implemented to calculate boundary conditions to add this information as inputs to the CFD vehicle thermal simulation.

Automakers have developed procedures to obtain the needed boundary conditions for normal aspirated gasoline engines. Nevertheless when heavy duty trucks or even small cars moved to diesel compression engine technologies in order to improve the fuel economy an trailer capacity, these procedures do not complete apply due the differences in the exhaust after treatment system for the emission control and the regeneration process used to burn the soot in the Diesel Particle Filter (DPF) system. In order to obtain the boundary conditions physical test and historical data are needed.

This work present the results of an investigation in order to create a simulation process combining 1-D and 3-D CAE tools for a diesel exhaust transient simulation during regeneration and a validation of the same procedure comparing the results of the simulation with a physical test data with the objective to improve the early stage of development for vehicles with diesel after treatments system.

This investigation implement a 1-D MATLAB SIMULINK regeneration model with a 3-D CFD FLUENT flow model for different steady state conditions. The model combines results obtained from a 1-D stream flow into a 3-D FEA RADTHERM model to have a complete transient thermal simulation.

2.-Nomenclature

CAE	Computer Assisted Engineering
DPF	Diesel Particle Filter
CFD	Computational Fluid Dynamics
C_s	Soot concentration
O_2	Oxygen concentration
CO_2	Dioxide of carbon concentration
$\frac{\partial T_{DPF}}{\partial t}$	Temperature in Diesel Particle Filter first differential
$m_{DPF,\eta st}$	Mass of soot in Diesel Particle Filter
$K_{DPF,st}$	Arrhenius Soot reaction rate
Ea	Activation Energy
R	Ideal gas constant
k_c	Reaction constant
ΔH_{st}	Soot's Entropy
$\dot{m}_{DPF,out}$	Outlet Mass Flow of exhaust gases in Diesel Particle Filter.
cp_{DPFgas}	Specific Heat of gases in Diesel Particle Filter
D'	Thermal Resistance
T_{DPF}	Diesel Particle Filter Temperature
$\dot{m}_{DPF,in}$	Inlet Mass Flow of exhaust gases in Diesel Particle Filter.

$T_{in,DPF}$	Diesel Particle Filter Inlet Gases Temperature
T_{Amb}	Ambient Temperature
D	Latent Energy due to Temperature
t_{st}	Stabilization test time

3.-Problem description

DPF is a device designed to remove Particle matter or soot from the exhaust gases, with the intention of meeting emission regulations for diesel engines.

DPF are also used to reduce the visible dark smoke from the exhaust gases, and as more soot is trapped in the filter it is necessary to clean it up in order to avoid high back pressure in the exhaust system or engine malfunction. This process is called regeneration

The most used method in the automotive industry is basically a soot catalysis activated by sending unburned fuel to the pipe line or by the direct injection of liquid urea into the filter [2].

DPF usually remove 80% of the particle matter in the exhaust gases and during heavy load conditions like regeneration it can remove 100%. It is during the regeneration when the exhaust gases achieve the maximum temperatures due the catalysis reaction in the filter.

In order to accomplish thermal safety regulations and product validation the DPF packaging in the underbody of the vehicle must be studied and understood to avoid possible thermal issues and minimize the impact to the nearest components. This is possible due the testing procedures development to learn the possible impacts in the under body.

For gasoline engines as for diesel engines this physical test process is common and the same test conditions apply for both [8].

This process can be simulated in CFD or CAE software but due to its complexity the diesel engine is treated different from the gasoline engine. The main difference is that for diesel engines skin temperatures in the exhaust components are used instead of exhaust gas mass flow, this approach tends to overestimate the underbody temperatures because all the pipe line is considered to be at a same temperature.

The main problem with this approach is that historical data o test data is needed in the early stages of the design process when it may not be readily.

To improve this process for diesel engines a combination of 1-D and 3-D, CFD and CAE transient simulation is development and validated to have a better understanding in the under body vehicle environment.

4.-Regeneration soot oxidation theory

Soot oxidation in the DPF can be managed as carbon oxidation reaction:

$$C_S + O_2 \xrightarrow{Heat} CO_2 \quad (1)$$

where C_S represent the combustible portion of the particle matter in the filter.

The rate of thermal regeneration is determined by the kinetic of soot oxidation using the Arrhenius law:

$$K_{DPF,st} = O_2 k_c e^{\frac{-Ea}{RT_{DPF}}} \quad (2)$$

Using GASEQ and the appropriate polynomials for Diesel fuel, it is possible determine the concentration of soot C_S in the exhaust gases during the stabilization. Also the enthalpy of the C_S could be computed.

After this it is possible to obtain the soot accumulation in the DPF:

$$m_{DPF,nst} = \dot{m}_{DPF,in} * C_s * t_{st} \quad (3)$$

Regeneration process can be defined by the following partial differential equation [1]:

$$\frac{\partial T_{DPF}}{\partial t} = \frac{-m_{DPF,\eta st} + K_{DPF,st} + \Delta H_{st}}{D} - \frac{(\dot{m}_{DPF,out}\, cp_{DPFgas} + D^{'})T_{DPF}}{D}$$

$$+ \frac{\dot{m}_{DPF,in}\, cp_{DPFgas} T_{in,DPF}}{D} + \frac{D^{'}T_{Amb}}{D} \qquad (4)$$

4.-Transient thermal simulation process

The validation thermal process used by the automakers can be replied with a 3 to 5°C of difference between test to test [8]. Depending of the vehicle sale region the conditions and the targets to accomplish can change but mainly the physical test has three stages:

1. Road load phase
2. Hill grade load
3. Idle

Boundary conditions for road load and idle phases of the test can be obtained with the common process for normal aspirated engines, since this part of the process does not involves regeneration. The hill grade load needs to involve the regeneration process that is induced during this phase of the test

The objective of this process is to obtain the increment in the exhaust gases temperatures due the regeneration reaction in the DPF, in order to use this as boundary condition for a 1-D stream flow inside in of 3-D CAE RADTHERM model and integrate the boundary conditions of all three stages of the test to run a complete transient simulation.

In order to capture the regeneration a CFD 1-D MATLAB SIMULINK model is implemented to solve the equation 4

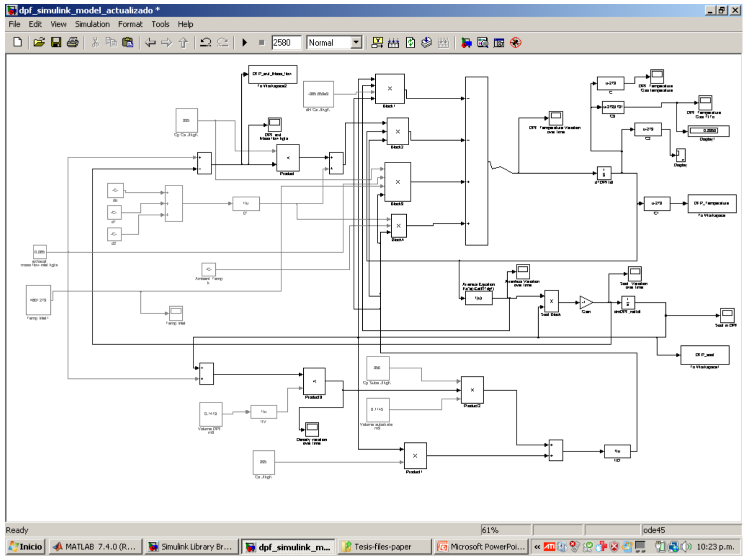

Figure 1. 1-D MATLAB SIMULINK model

The output of the 1-D model is shown figure 2, the temperature data was normalized using the maximum skin manifold temperature in order to present them in this article.

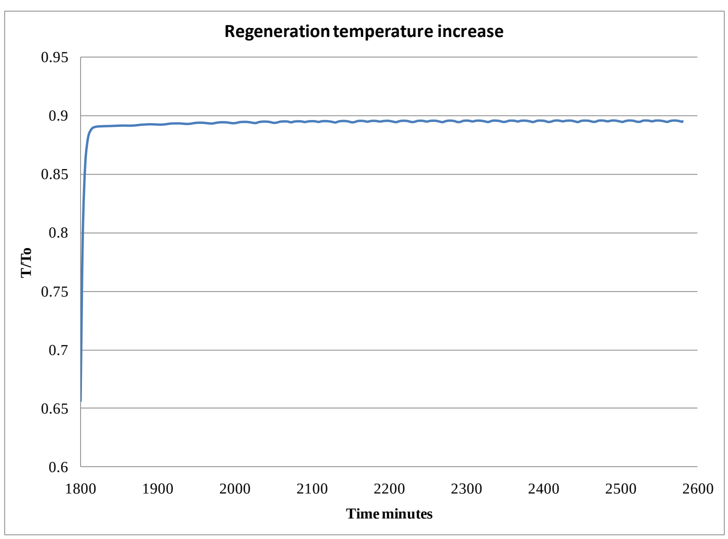

Figure 2. 1-D output T/To vs. time during regeneration.

Now it is possible to have the after treatment mass gas flow temperature during the thermal test procedure with regeneration, once the boundary conditions are acquired a 3-D mesh model of the exhaust system is created in fine TRIA elements and is integrated in a

complete underbody vehicle CFD 3-D flow model. The same approach in gasoline engines is followed here considering gas mass flow inside the pipe line after the turbo stage.
Flow thermal simulation in steady state conditions are run using CFD software in order to compute the convective values of the surrounding environment using previous computed boundary conditions. As a result of the steady state simulations, fluid temperature and fluid convection values are computed, to run a complete transient simulation this values are exported in a generic format in order to use them as boundary conditions for the different stages of the test.

It is needed to run a steady state case for each of the test phase's previous described, figure 3 shows the CFD Heat transfer coefficient obtained from the steady state simulation using FLUENT as solver.

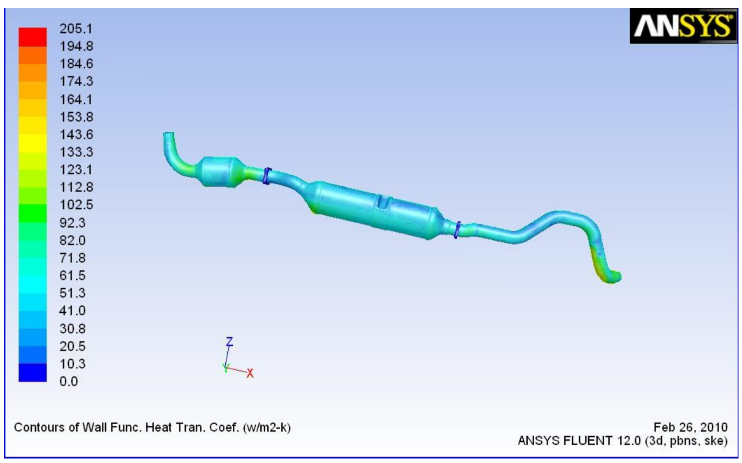

Figure 3. 3-D steady state convection heat transfer coefficients for exhaust during regeneration

The exhaust parts and any other under body part that want to be evaluated is re-meshed from the fine TRIA elements model into a coarse QUAD mesh in order to reduce the elements size and optimize the solution time and heat transfer results.

This model is imported to RADTHERM, where the results of the steady state solution are imported to the surfaces of the QUADs model and the fluid data is mapped into the quad elements. This is done at the beginning time and the end time of each test phases in order to incorporate these data to the transient simulation.

The last step is to create a 1-D flow stream part inside the exhaust pipe line to incorporate the mass flow rate and the gases temperatures to acquire the complete thermal inertia of the exhaust system during the duration of the test.

5.-Results

After the integration of all the steps development in this methodology:

1. 1-D SIMULINK model to compute the regeneration gas temperature
2. 3-D FLUENT vehicle model with mass flow gases to compute the convective temperatures and heat transfer coefficients at steady state.
3. RADTHERM 3-D QUADs elements model with 1-D flow stream to and imported steady state conditions.

The transient simulation is performed; the output of the process is a complete transient thermal simulation.

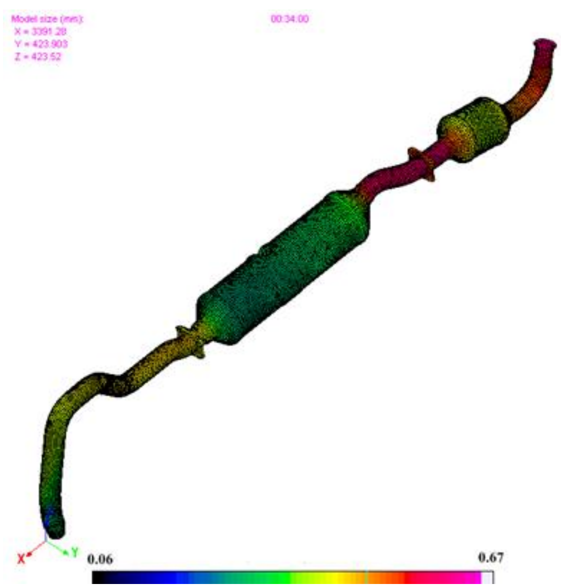

Figure 4 Exhaust Temperature contour (T/To) at 34minutes

Figure 4 is an example of the output obtained from the transient simulation in which we can observe the different temperatures contours of the after treatment system after 34 minutes, the results for the DPF body part of the exhaust system and a comparison with the physical test in order to observe the correlation achieved by the implementation of this simulation process can be observed un figure 5.

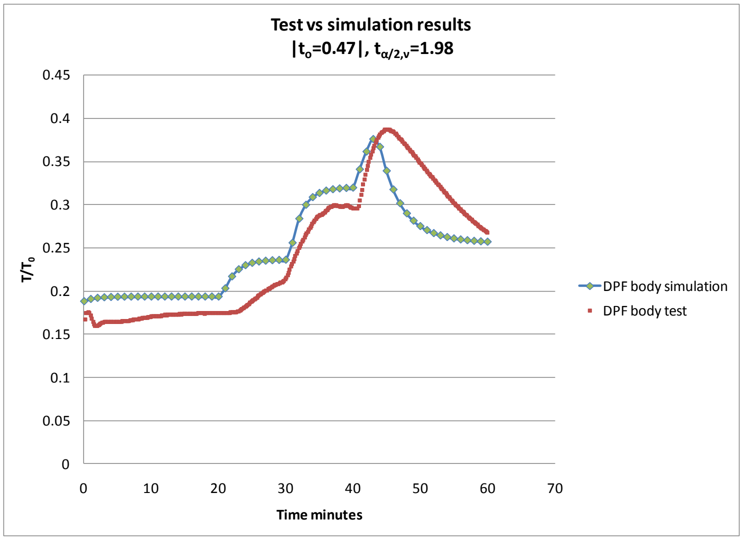

Figure 5 DPF Body

In figure 5 is observed that the trend of the temperature rise is similar between the test and the simulation during the hill portion of the test, in the final part of the test were the idle start the thermal inertia is well capture during the initial part of the idle, this phenomenon occurs due the beginning of the cold down of the exhaust mass flow gases and a decrease of the convection heat transfer coefficients, finally it is observed that the thermal inertia in the latest part of the idle is longest in the test than in the simulation having an offset of 10 minutes however the temperature ranges are under the same values and finally archiving the same final temperatures at the end of the test.

6.-Conclusions

During Advance vehicle development process this procedure is a value adds for new diesel engines with DPF technology, when data is needed to make decisions and not enough test data is available.

And advantage of the simulation is the results can be computed for all part of the after treatment system and can be measured at any point of interest and changes to the system and the packaging can be observed in short time solving possible issues in an early stage of the design process.

The 1-D MATLAB SIMULINK model captured correctly the temperature increment during regeneration this allowed to have accurate boundary conditions that were used in the next steps of the simulation process.

The process that included the integration of several software outputs to complete a transient simulation, proved to capture a proper temperature skin distribution of the exhaust system during the simulation. The data compared with the physical test using student T distribution with α=95% for the mean values, were under the acceptation ranges, this for every part of the DPF demonstrating that the simulation results have the same trend as the physical test.

The thermal inertia for the idle part of the test was capture with an acceptable trend by the simulation. This is a direct result of the use of the 3-D flow thermal steady state convection values obtained with the CFD software. Otherwise these values would not represent accurately this effect.

This simulation process was correlated and validated for a V8 diesel engine future work includes testing the process with V6 and L4 diesel engines.

One of the main limitations of the current process is that the pressure drop in the exhaust system is not considered so not back pressure results could be obtain from this process.

The integration of this methodology to the design process is in progress for the virtual vehicle development process in order to predict under body temperatures during regeneration.

7.-Acknowledgments

Special recognitions to the following people that help in the elaboration of this study:

- David Martin HVAC/PTC/Thermal Director.
- Jorge Estevez GMM HVAC/PTC/Thermal Supervisor.
- Edgar Vega Ambia GMM HVAC/PTC/Thermal Supervisor.

- Frank Miller GMM HVAC/PTC/Thermal Director retired.
- Dr. José Carlos Miranda ITESM Toluca DIA Director.

8.-References

[1] Su F., Gordon B.W. and Hong H. *Simulation of a Diesel Engine After treatment System Using Singularly Perturbed Sliding Manifold* (2007).

[2] Hinterberger C., Olesen M. and Kaiser R., 3D *Simulation of Soot Loading and Regeneration of Diesel Particulate Filter Systems* (2007).

[3] Konstandopoulos A. G., Kostoglou M., Vlachos N. and Kladopoulou E. *Progress in Diesel Particulate Filter Simulation* (2005).

[4] Konstandopoulos A. G., Kostoglou M. and Lorentzou S. *Wall-scale Reaction Models in Diesel Particulate Filters* (2007).

[5] Masoudi M. Konstandopoulos A. G., Nikitidis M. S., Skaperdas E., Zarvalis D., Kladopoulou E. and Altiparmakis C. *Validation of a Model and Development of a Simulator for Predicting the Pressure Drop of Diesel Particulate Filters* (2001).

[6] Montenegro G., Piscaglia F., Ferrari G. and Onorati A. *A 1D Unsteady Thermo-Fluid Dynamic Approach for the Simulation of the Hydrodynamics of Diesel Particulate Filters,* (2006).

[7] Wirojsakunchai E., Kolodziej C., Yapaulo R. and Foster D. E. *Development of the Diesel Exhaust Filtration Analysis System (DEFA),* (2008).

[8] General Motors *Thermal development process,* (2008).

[9] Vlachos N., Patrianakos G., Kostoglou M. and Konstandopoulos A. G. *Micro-Simulation of NO-NO2 Transport and Reaction in the Wall of a Catalyzed Diesel Particulate Filter* (2008).

[10] *Thermodynamics and engineering approach,* Yunus Cengel. Ed McGraw Hill 6th edition.

[11] *Motores de combustion interna análisis y aplicaciones,* Orbert Edward, Ed. Compañía editorial continental.

[12] John. H. Johnson *Diesel Particle Emissions Landmark Research* 1994-2001, SAE.