Simulation of Exhaust Gas Heat Recovery System for an Automobile

Anil Kumar Jaswal[*1], David Neihguk[2], and Abhinav Prasad[3]
[1]Mahindra & Mahindra Ltd.
*Mahindra Research Valley, Mahindra World City, Chennai, Tamil Nadu, India
jaswal.anil@mahindra.com

Abstract: This paper presents the simulation of exhaust gas heat recovery system for an automobile using COMSOL. A double pipe heat exchanger is modeled in which the tube side medium is the hot exhaust gas and the shell side medium is water. The problem is divided into three parts i.e. Laminar flow in water regime, Turbulent flow in Hot exhaust gases and Heat Transfer in solids. Turbulent flow is solved using k-ε turbulence model. They are solved independently and the results are coupled subsequently. Different cases are run to validate the experimental data for different time steps. The Laminar flow problem and Turbulent flow problem are solved simultaneously taking the flow to be steady state in order to simply the problem. Whereas, Heat Transfer in solids physics in COMSOL library is used to account for the time-dependent nature of the problem. For meshing, user controlled mesh is used and free tetrahedral meshing is implemented on all the domains. The results are found to be quite satisfactory and shows a good correlation with the experimental data.

Keywords: Heat Exchanger, Heat Transfer, Fluid flow and COMSOL Multiphysics.
.

1. Introduction

A parallel flow concentric tube heat exchanger is designed to recover exhaust heat by using the exhaust pipe as the tube side and the concentric tube as the shell side. The heat exchanger also called primary heat exchanger, is welded concentrically to the intermediate exhaust pipe forming a double pipe heat exchanger. The shell side pipe consists of Cold Water inlet and Hot Water outlet while the tube side pipe carries the hot exhaust gases. The water flowing in the shell side gets heated due to conduction and convection and is circulated through the Hot Water outlet to produce Hot Water on demand. The same Hot Water is simultaneously circulated to the exterior section of an annular chamber called Hot Box which is also called the secondary heat exchanger, thereby transferring the heat to the interior section.

However, modeling of the secondary heat exchanger is not included in the present analysis.

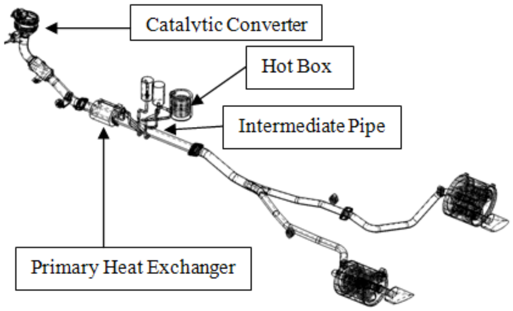

Figure 1 : Schematic diagram of Exhaust gas heat recovery system

2. Model

Figure. 2 shows the schematic diagram of the heat exchanger. The top side is the entry for the circulated water from which the heat has been extracted at the Hot-Box. The outlet is located on the downside of the heat exchanger so that hot water can be easily pumped out of the heat exchanger.

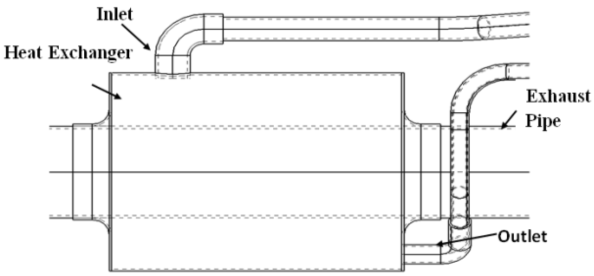

Figure 2 : Schematic diagram of Primary Heat Exchanger

The heat exchanger is made of stainless steel. The heat exchanger medium is water which is flowing through the shell side and the hot exhaust gas flows through the tube side. The thermal conductivities at 25 deg C of the heat exchanger material and the fluid medium are given in Table 1.

Table 1: Thermal conductivities

Substance	**Thermal conductivities ($\mathbf{Wm^{-1}K^{-1}}$)**
Stainless steel	16
Water	0.58
Exhaust Gas (Carbon dioxide)	0.0146

A heat transfer of 15 W/m^2K is in this simulation.

This model uses the Non-Isothermal Flow user interface together with the k-ε turbulence model. It takes advantage of symmetries to model only one half of the heat exchanger, thereby reducing model size and computational costs.

2.1 Assumptions

The following assumption have been taken into consideration:

(1) The fluid is considered to be incompressible.

(2) The flow in the water regime is assumed to be streamline and laminar.

(3) No slip boundary condition is assumed in the laminar regime.

(4) Symmetry is assumed therefore only half of the section have been simulated.

2.2 Governing equation

Based on the model assumptions, the flow field of water regime is governed by the continuity equation which insures the mass conservation and the steady state incompressible Navier-Stokes equation which describes the momentum conservation of Newtonian fluids.

Navier-Stokes Equation:

$$\boldsymbol{\rho}(\mathbf{u}.\nabla)\mathbf{u} = \nabla.[-\mathbf{PI} + \mathbf{v}(\nabla\mathbf{u} + (\nabla\mathbf{u})^{\mathbf{T}})] + \mathbf{F}$$

Continuity Equation:

$$\nabla.\mathbf{u} = \mathbf{0}$$

where **u** is velocity vector, m/s; P, pressure, Pa; **F**, body force, N/m^3; $\boldsymbol{\rho}$, density, kg / m^3 ; **v**, kinetic viscosity, m/s^2, ∇ is the vector differential operator.

The governing equation for heat transfer in the model is the heat equation for conductive and convective heat transfer.

$$\nabla.(-\mathbf{k}\nabla\mathbf{T}) = \mathbf{Q} - \boldsymbol{\rho}.\mathbf{Cp}.\boldsymbol{\mu}.\nabla\mathbf{T}$$

The temperature dependent properties for water, air and metal are taken from the built in library.

So the equation for the heat transfer in solids is given by,

$$\boldsymbol{\rho}.\mathbf{Cp}.\boldsymbol{\mu}.\nabla\mathbf{T} = \nabla.(\mathbf{k}.\nabla\mathbf{T}) + \mathbf{Q}$$

3. Boundary conditions

The following boundary conditions are used for solving the model:

(1) k-ε turbulence equation in the gas domain:
  a. Specified initial velocity.
  b. Specified initial temperature.
  c. Symmetry at the boundary region.
  d. Wall function at the pipe and gas region.
  e. Fixed outlet pressure.

(2) For water regime:
  a. Specified initial velocity.
  b. Specified initial temperature.
  c. Symmetry at the boundary region.
  d. No slip condition at wall.
  e. Fixed outlet pressure.

4. Meshing:

The mesh plays a very vital role in solving a fluid flow model. Accuracy of the solution is defined by the mesh size only but it doesn't mean that the accuracy will directly depend on the mesh size. After reaching to a specific point results become independent of the mesh size i.e. on further increase in the mesh size will not affect the results.

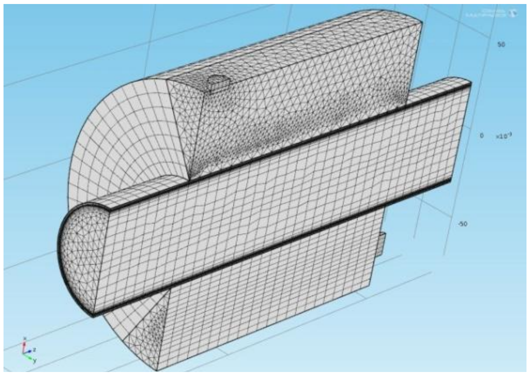

Figure 3 : Meshed Model

Figure 3 shows a meshed fluid flow model. On most of the locations free triangular mesh have been used. On some locations free tetrahedral mesh is also used. On the inlet of exhaust gas triangular mesh is used and it is swept along the length. A boundary layer mesh is also applied in the normal direction of pipe boundary. It is created by inserting structured layers of elements which are very helpful to solve fluid flow applications coupled to mass and energy transfer.

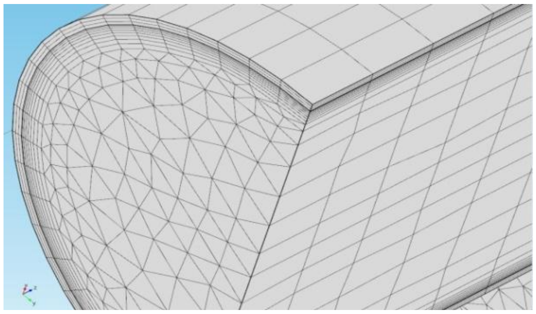

Figure 4 : Boundary layer mesh

The details of meshing is given in Table 2

Table 2: Mesh statistics

S. No.	Property	Value
1	Minimum element quality	0.003193
2	Number of elements	37443
3	Average element quality	0.439
4	Tetrahedral elements	11889
5	Prism elements	11210
6	Triangular elements	3055
7	Edge elements	805

5. Results and Discussions

Taking the results obtained from measurement, Table 3 shows the maximum temperature during the test which is done in highway driving condition.

Table 3. Results of Dynamic testing at Highway Driving

^{o}C	**Start**	**End**
T1	36	348
T2	38	336
T3	31	33
T4	36	94

where

T1	Exhaust gas inlet temperature
T2	Exhaust gas outlet temperature
T3	Inlet water temperature
T4	Outlet water temperature

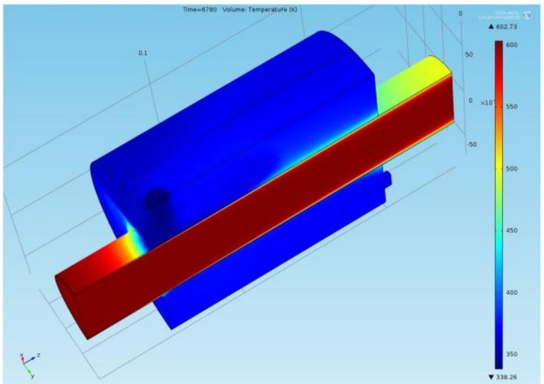

Figure 5 : Temperature distribution contour

The inlet temperature and inlet velocity for the water domain are given as 32 ºC and 0.17 m/s respectively. For the exhaust gas side the inlet velocity is given as 37 m/s and the inlet temperature at the inlet is given as 117 ºC.

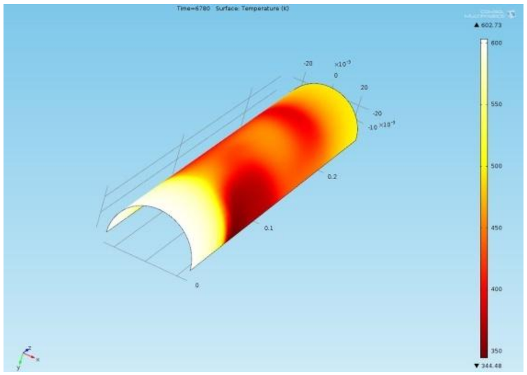

Figure 6 : Temperature distribution in exhaust pipe

In figure 5, we can see the distribution of temperature in the water regime as well as in the exhaust gas. The temperature at the inlet of the water is less in comparison to the temperature at the other locations of the domain. There is a sudden rise in water temperature at the contact area mainly due to heat transfer by conduction and convection. Also, it can be seen that there is a gradual drop in the gas temperature in the axial direction as the heat is transferred from the exhaust gas to the water.

The corresponding heat loss from the exhaust pipe in the axial direction can be observed from figure 6.

The outlet temperature of hot water is plotted against time. Figure 7 (Appendix) shows the comparison of experimental and simulation results with respect to the rise in water temperature. The sudden dip in experimentally obtained data at about 3100, 4000 and 5000 seconds is due to addition of cold water in the heat exchanger system. Whereas, in COMSOL simulation, the outlet water temperature is given as the input for water inlet. Therefore, the variation of outlet gas temperature is comparatively smooth compared to the experimental data.

7. Conclusions

The use of COMSOL Multiphysics to perform preliminary analysis of heat transfer mechanism in Exhaust gas heat recovery system is found to give good correlation with the testing results.

8. References

1. David Pumthianlal Neihguk; Aravindh Rajaram; Samir Rawte, Non Intrusive Exhaust Gas Heat Recovery System for an Automobile: An Experimental Investigation, *SAE 13CVI-0097* (Manuscript under review)
2. Kakaç, Sadık, Anchasa Pramuanjaroenkij, and Hongtan Liu. Heat exchangers: selection, rating, and thermal design. CRC press, 2012
3. Incropera, Frank P., Adrienne S. Lavine, and David P. DeWitt. Fundamentals of heat and mass transfer. John Wiley & Sons Incorporated, 2011

9. Acknowledgements

The authors acknowledge the support given by COMSOL India.

10. Appendix

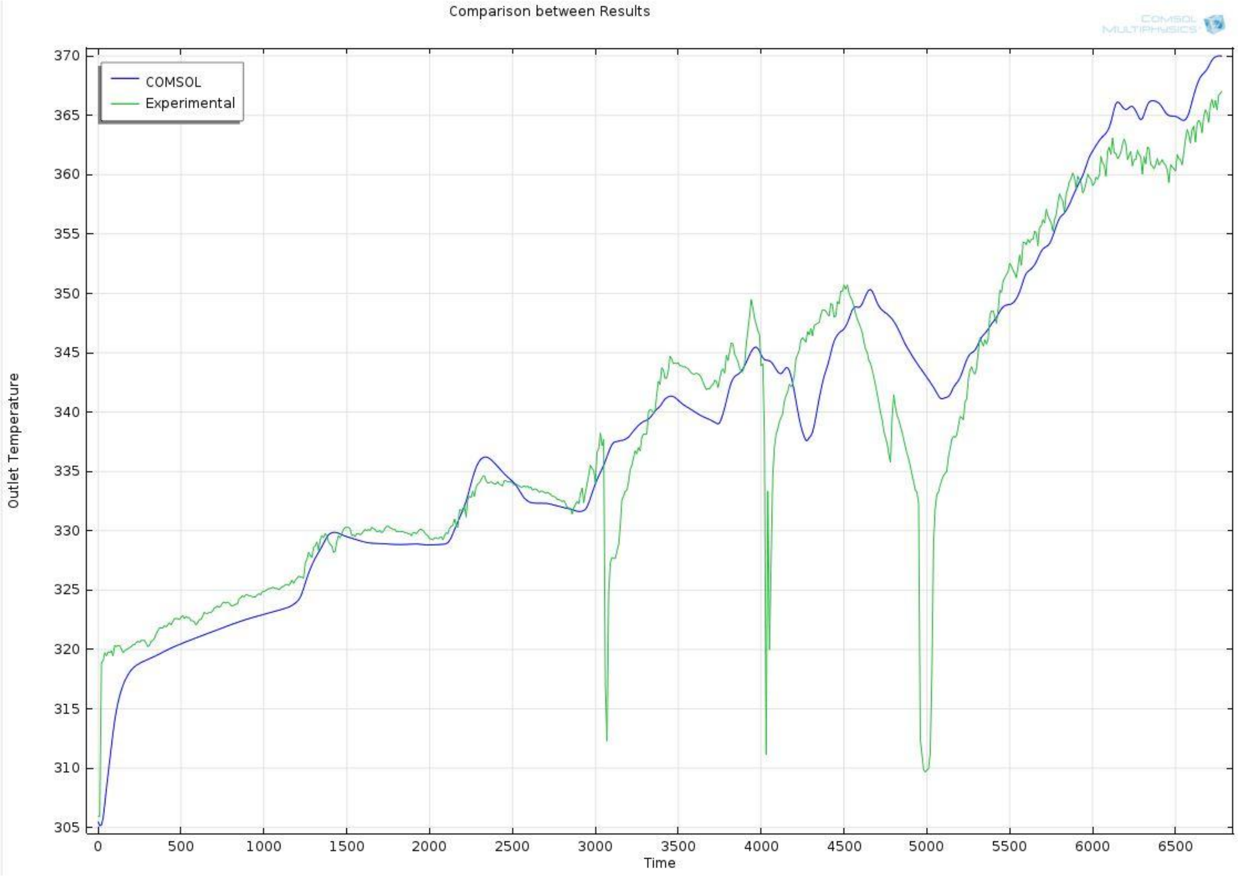

Figure 7 : Outlet water temperature COMSOL Vs Experimental