International Journal of Engineering Inventions
ISSN: 2278-7461, ISBN: 2319-6491
Volume 2, Issue 1 (January 2013) PP: 19-21

Simulation of the Design of an Exhaust Silencer Stack by CFD

H.Maheshappa[1], V.K.Pravin[2], K.S.Umesh[3], P.H.Veena[4]
[1]Dept. Of Mech. Engg. ; Vice Chancellor VTU Belgaum, Karnataka,India
[2]Dept. Of Mech. Engg. ; P.D.A College of Engg, Gulbarga, Karnataka,India
[3]Dept. Of Mech. Engg. ; Thadamal Sahani College of Engg. Bombay, Maharastra, India
[4]Dept. Of Mathematics.; Smt.V G College for Women, Gulbarga, Karnataka,India

Abstract:- A concern during stack emission testing is the sampling port location. The sampling port must be selected so that the velocity measurements truly represent the exhaust gas flow field within the exhaust stack. Usually the port is located near the stack exit. In some instances during field testing, however, concerns have been raised over the location of the port and its influence in accurate measurements. Furthermore, the use of perforated baffles in the silencer stack has also been questioned; it has been suggested that these could cause flow disturbances that are not acceptable. The veracity of the concerns raised can be examined by physically changing the baffle design and the location of the port. However, this approach would not only take an extensive period of time but also would prove expensive. A lower cost approach is to conduct a CFD analysis on the existing silencer stack with CFX software and then analyze the results to determine the impact of small perturbations on the measurements and the overall flow field. Modifications, if needed, could be made in the solid model of the silencer/stack system. The impact of these modifications could then be tested with the CFD model to determine how the unmodified stack compares to the modified stack and to establish the degree to which the baffle perforations and the port location Influence testing results. This paper describes a detailed CFD analysis of the flow pattern in the silencer stack.

I. INTRODUCTION

The impact of the silencer baffles on emission stack testing continues to be a controversial issuebetween operator's emission engineers, and regulators. While emission testing of a relativelyhomogeneous mixture appears to be fairly straightforward, some argue that measurement may be complicated not only by the presence of silencer baffles but also by port location, portgeometry, and atmospheric air flow at the sampling port. Since the impact of these factors is not fully understood, operators and environmental engineers may compensate by increasing the emission operating margin. The use of perforated baffles in the silencer stack has also been questioned. it has beensuggested that these could cause flow disturbances that are not acceptable. The veracity of theconcerns raised can be examined by physically changing the baffle design and the location of the port. However, this approach would not only take an extensive period of time but also would prove expensive. A lower cost approach is to conduct a CFD analysis on the existing silencer stack and then analyze the results to determine the impact of small perturbations on themeasurements and the overall flow field. Modifications, if needed, could be made in the solidmodel of the silencer/stack system. The impact of these modificatons could then be tested withthe CFD model to determine how the unmodified stack compares to the modified stack and toestablish the degree to which the baffle perforations and the port location influence testingresults. An analysis based on virtual prototyping Through CFD can therefore, be an asset in the design cycle if used with appropriate modeling strategies.

II. CONSTRUCTION

A typical absorptive silencer contains passages that are lined with porous sound absorbing material and then shielded with a protective covering, such as metal, to protect it from the ambient air flow. Figure shows the solid model geometry of a typical silencer. Flow enters the inlet and bifurcates into channels that are lined with sound-absorbing porous material and exits into the atmosphere. The type of porous material used is dependent upon the application but may include mineral wool, fiberglass or foam in appropriate densities and thicknesses. Figure illustrates the wire frame geometry of the silencer. The silencer was 19.25 ft long. The channels were15 ft long, 6.5 ft wide and 0.65 ft deep. The porous material between the two channels was 1.5 ft thick while that above and below the channels were only 1 ft thick.

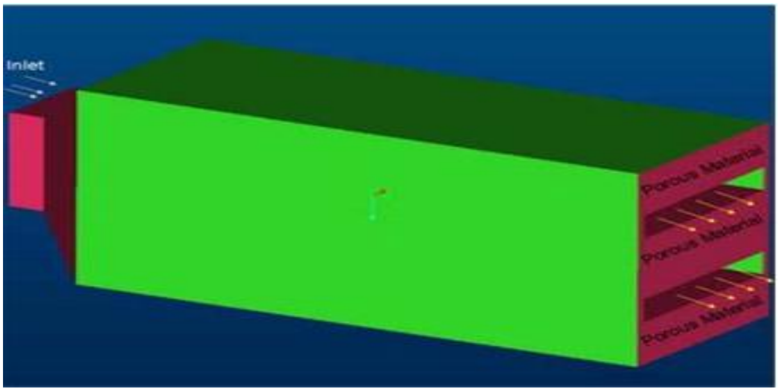

Figure 1: Geometric model of the Exhaust Silencer Stack

III. DOMAIN CONDITIONS

Single phase single species simulation will be carried out under isothermal conditions for air as the material. Turbulence will be modelled by k-ε RNG turbulence model appropriate to account for high velocities and strong streamline curvature in the flow domain. The reference pressure will be set at 1 atm and all pressure inputs and outputs will be obtained as gauge values with respect to this.

BOUNDARY CONDITIONS

The boundary conditions used to solve for the velocity and pressure distribution throughout the channels were:
Inlet: Mass flow rate = 10 kg/s
Outlet: Pressure = 74,000 Pa

RESULTS AND DISCUSSIONS

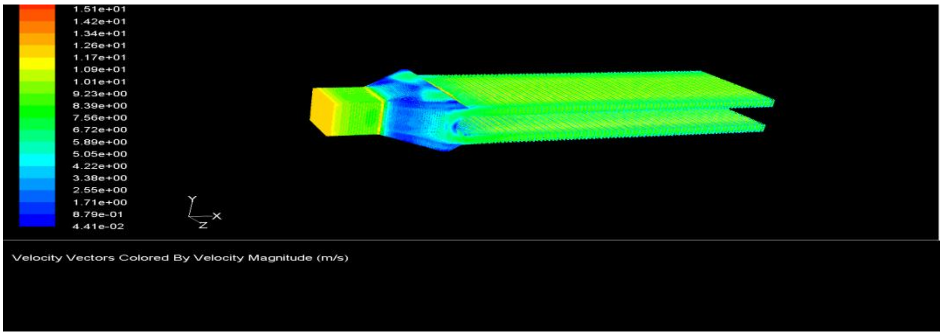

Fig.2 velocity contour

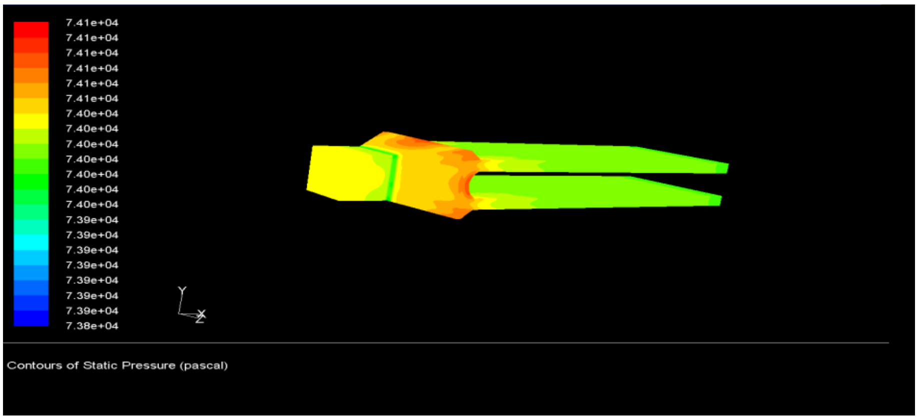

Fig.3 pressure contour

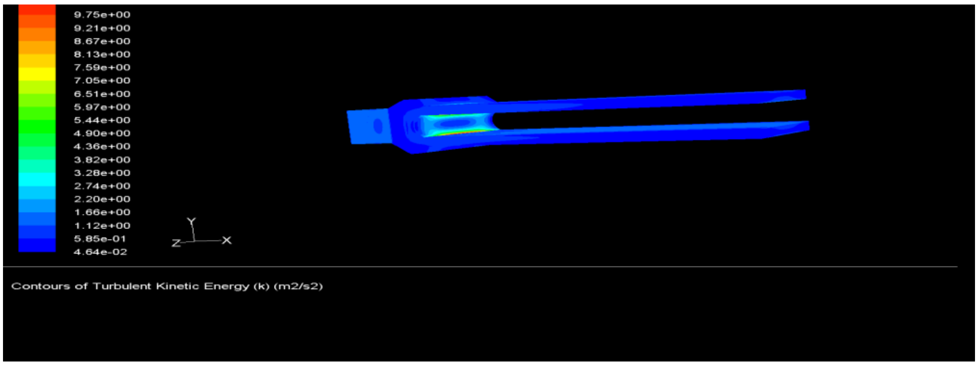

Fig.4 Turbulent kinetic energy

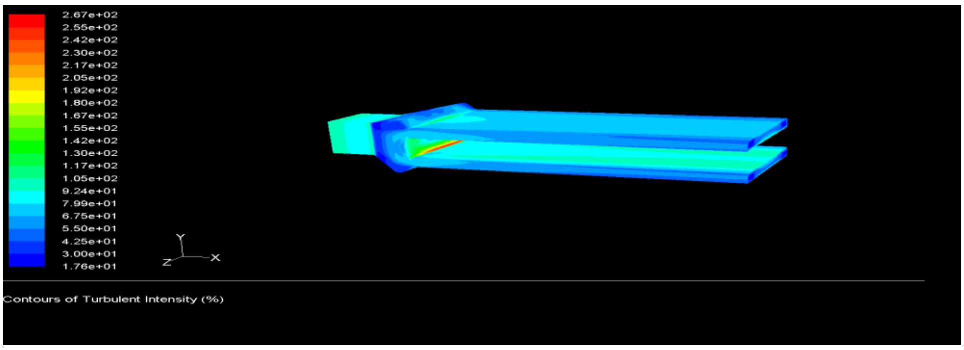

Fig.5 Turbulent intensity

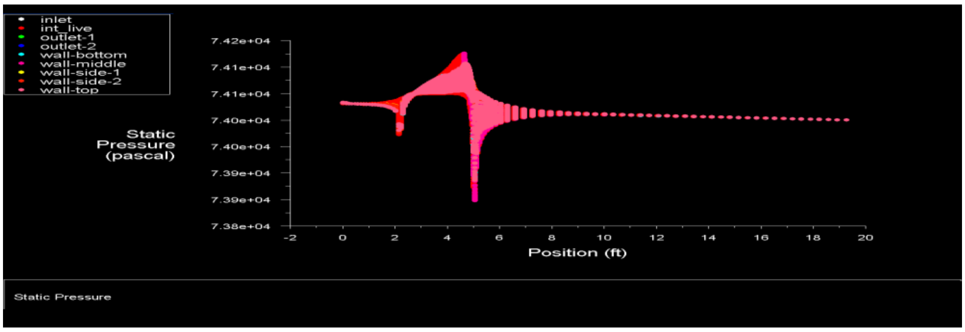

Fig.6 Pressure V/s position of stack

IV. CONCLUSION

The velocity flow at the exhaust (outlet) in the case of the silencer with porous is reduced when compared with the non porous silencer stack The pressure distribution for the silencer with porous is not uniform along the stack but in the case of non porous silencer stack it is uniformly distributed The average pressure drop in the porous condition is high as it is due to the flow friction this not the case in without porous condition

REFERENCES

1) Bird, B. R., W. E. Stuart and E. N. Lightfoot, 2002, *Transport Phenomena*,.
2) CFX, 2004, *Solver Theory*, CFX 5.7.1 Users Manual, Ansys, Canada.
3) Chapman, K.S., and A. Keshavarz, 2003, “Development of Turbocharger-Reciprocating Engine
4) Computer Simulation (T-RECS), Gas Research Institute, GRI-03/0005.
5) White, F. M., 2003, *Fluid Mechanics*, McGraw Hill, NY.