Using Virtual Engineering Techniques to Aid with Design Trade-Off Studies for an Enclosed Generator Set

Richard Gault[a,1], Richard Cooper[a], Jian Wang[a], Srinivasan Raghunathan[a] and Graeme Mawhinney[b]

[a]School of Mechanical and Aerospace Engineering, Queen's University of Belfast, Northern Ireland.
[b]FG Wilson Engineering Ltd.

Abstract. The influence of even a single parameter change can be difficult to second guess at the design stage of a product. There may be many fluid, acoustic or structural performance effects that are sensitive to specific parameters. Modifiying the product to satisfy one area of performance may weaken it in others. The sensitivity of parameter changes in terms of performance can be extremely difficult to determine through experimentation. Virtual Engineering techniques were used to highlight the sensitivity of Sound Pressure Level (SPL) and cooling airflow rate for a typical inlet vent of a power generator set, whose open area was reduced to induce a parameter change. For this parameter change, Computational Fluid Dynamics (CFD) simulation showed how the cooling airflow rate was unaffected. An acousitc analaysis using the engine and alternator as the source noise showed how the Sound Pressure Levels (SPLs) were sensitive to this parameter change at low frequencies. This work underlines the necessity to always consider both the acoustic and cooling performance as a coupled system during design for this particular product.

Keywords. Virtual Engineering, CFD, Multi-physics, Acoustics, Enclosure

1 Introduction

Power generator sets are used to provide secure electric power for a vast arena of applications, ranging from prime power in remote locations and developing regions to construction power, standby power and emergency power for the grid. Traditionally, designs are generated based on previous experience and empirical algorithms, which usually involves an iterative modification process on a specially developed prototype. With ever increasing global competition, methods for synthetically modelling new products are essential to reduce lead-time, product

[1] Research Fellow, School of Mechanical and Aerospace Engineering., Ashby building, Stranmillis Road, Belfast, BT9 5AH, Northern Ireland; Tel: +44 (0)28 9097 5642; E-mail: r.gault@qub.ac.uk

cost and remain within noise legislative targets. Virtual Engineering techniques provide a platform to evaluate design trade-off studies prior to design implementation. This paper uses two virtual engineering methods applied to a diesel engine generating set to evaluate the trade-off between cooling airflow and noise for a single parameter change. In general the cooling airlow is usually induced using a cooling fan driven by the engine. Cluttered environments, including various types of inlet/outlet louvres, ducts, bends and various cowls etc., can significantly affect the system pressure loss resulting in the fan performance being impared.

The major elements in the total sound energy radiation from a diesel generating set are due to the action of its vibrating solid surfaces and aerodynamic noise [1]. The presence of ventilation openings significantly weakens the overall sound attenuation characteristic of the enclosure. Frequently, generator sets are installed in areas of high sensitivity where the acceptable noise levels are very low, especially as mains power can fail at any time, day or night. The control of diesel engine noise presents a number of particular problems and also offers scope for a number of treatments designed to give particular benefits to the users. The growing environmental awareness has led to recently tightened European regulations relating to the permissible sound power level of power generating sets.

Prediction of noise from a cluttered enclosed diesel generator set is inherently complex due to geometric shape complexities and variable damping factors of the system components including complex sound transmission mechanisms through multi-layered structures. Further, the sound radiated by the individual sources is influenced by the rest of the machine altering the radiation directivity. Several noise tranmission paths are prevalent, namely the airborne path, mechanical path and coupled sound transmission path. Ju et al. [2] used boundary element methods and CFD to aid with the design of an acoustic enclosure for their application. Interestingly they measured the source noise using a sound intensity method to calculate the sound power then used this as a boundary condition to drive the source model. Choi, Kim and Lee [3] also considered a simple design method of the engine enclosure considering cooling and noise.

2 Virtual Engineering Technique for Airflow Analysis

2.1 CAD Modelling and Mesh Generation

Using a bottom-up topological approach, the surfaces of a typical generator set, namely air inlet vents, engine, alternator, axial cooling fan, radial cooling fan, shroud and outlet box, were generated. Detail like bolt holes, ribs, stiffeners etc., were not modelled as the cost numerically, both in generating the mesh and running the solver, would be too great. As both the axial and radial cooling fans are inducing the airflow, their blade profiles were generated through a process of reverse engineering using point cloud data from a series of 3D laser scans on the actual blades themselves. Both the axial and radial fans can be seen in Figures 1(a) and (b) respectively.

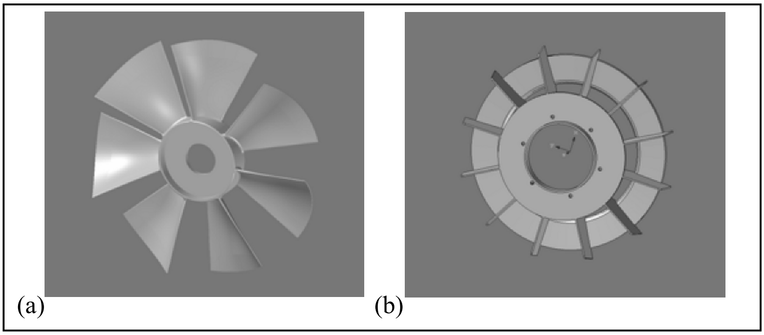

Figure 1. CAD representation of cooling fans generated using reverse engineering techniques for (a) axial fan and (b) radial fan.

2.2 Meshing, Pre-Processing and CFD Analysis

To minimise turnaround time an unstructured mesh was generated. The mesh was typically in the order of 2.5million elements including tri, tetra and prismatic elements. Prismatic elements were grown from the fan blade surfaces to capture the boundary layer. Figure 2 (a) shows the surface geometry of the enclosure.

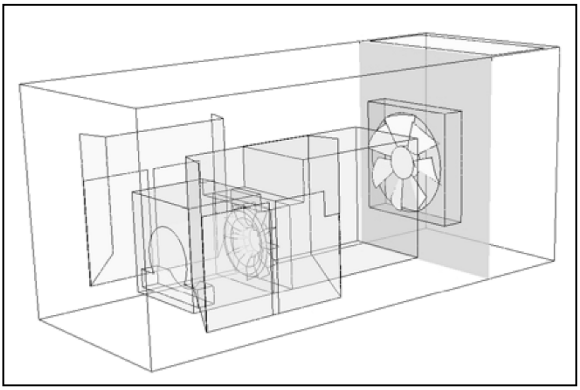

Figure 2. Generator set CAD model.

The commercial code Fluent [4] was used for the flow simulations. The Reynolds Averaged Navier Stokes (RANS) equations were solved and the flow was assumed to be an incompressible gas. Inlet vent boundaries were imposed on the ventilation openings, along with a nominal turbulence intensity of 1% and length scale of 0.08m. Polynomial loss coefficients, determined from experimental data, were imposed on the inlet boundaries. An outlet vent boundary was used at the exit of the enclosure, with the same imposed loss polynomials as the inlets. A multiple rotating reference frame model was used to simulate the rotation of both fans. Adiabatic no-slip walls were used elsewhere in the domain. The standard *k-e* turbulence model was used to close the RANS equations. Second order discretisation schemes were used for the governing flow equations. In order to model a restriction caused by the radiator, loss coefficients were determined from experimental values of the radiator core pressure drop. Standard atmospheric conditions were assumed.

Typically after 2500 iterations, a converged steady state solution was obtained as evidenced by the residuals flat-lining and by the drag monitors on the rotating fans.

2.3 CFD Results

The flow rate was predicted at 1.92m^3/s, for the initial total inlet area of 0.92m^2. The inlet vent normal face velocities varied from approximately 0.1m/s to just over 6.0m/s around the top of the inlets, Figure 3(a). The velocity profile exists due to the nature of the flow into the duct in this particular case. For the parameter change, the total inlet area was reduced to 0.52m^2, in effect blocking off the portion of the inlet where the normal face velocities were low. The flow rate for this new case was identical to the first computation at 1.92m^3/s. The normal to face velocities are all now approximately between 5.0m/s and 6.5m/s. What this suggests is that the open area can be reduced by approximately one third of the original area without any penalty in cooling airflow rate. Also in this instance the pressure drop in the enclosure was similar for both cases.

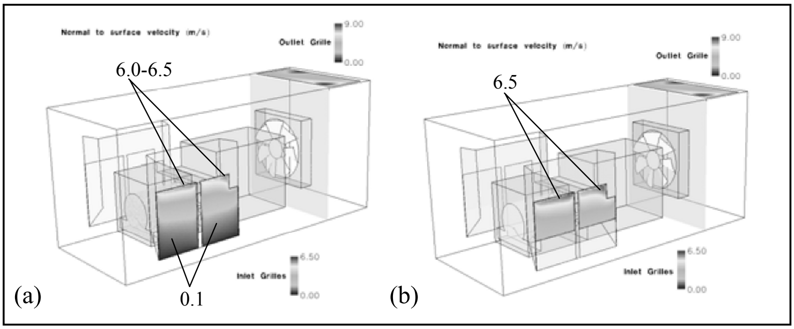

Figure 3. Normal to face velocities for (a) original inlet vent and (b) reduced area inlet.

3 Virtual Engineering Technique for Airborne Noise Analysis

In order to predict the SPLs at the ventilation openings for the same generic CAD model as that detailed earlier, the multi-physics solver RADIOSS [5] was used. Multi-physics is becoming an ever increasing virtual engineering method and involves the modelling of fluid/structure interaction as a coupled system. In effect

it brings together these separate disciplines, enabling the user to model the problem in a more realistic way. RADIOSS is an explicit finite element code, solving the compressible fluid flow equations with solid structures, using an Arbitrary Lagrangian Eulerian formulation (ALE) [6].
For this solver a 3 million element structured grid was generated, adhering to specific mesh sizing criteria in the noise generation and propagation zones.

3.1 Pre-Processing

The Young's modulus, Poisson ratio and density were defined for the material properties of the various components of the enclosure. A fluid/structure interface was defined between the solid surface mesh and the surrounding fluid mesh. Lagrangian x, y and z boundary conditions were defined for the fluid in contact with the solid walls. Non-reflecting boundaries were defined at the ventilation openings.

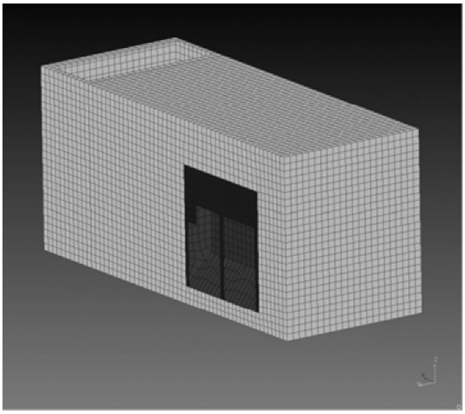

Figure 4. Global mesh structured grid representation of the generator set for the acoustic analysis.

3.1.1 Engine/Alternator Noise Source Characterisation

In order to mimic the noise generation from the vibrating engine and alternator surfaces, a detailed experimental survey of the surface vibration was undertaken using Laser Doppler Vibrometry (LDV). With the instrumentation, vibration data over a frequency band ranging from 50Hz up to 1000Hz was measured. Spacing of measurement points on the engine/alternator surface were 1/6th of the maximum wavelength of interest, in this case approximately 0.06m, avoiding any severe forms of spatial aliasing. For all the surfaces of the engine and alternator, approximately 700 measurement points were required to adhere to the spacing criterion. The sample rate was set at 4kHz for sample duration of 1.0 second. Figure 5 shows a typical vibration time history for one of the measurement locations on one side of the engine/alternator. A reference vibration on the bore of the engine was used to ensure that all the subsequent measurements on the various surfaces were phased locked relative to the reference.

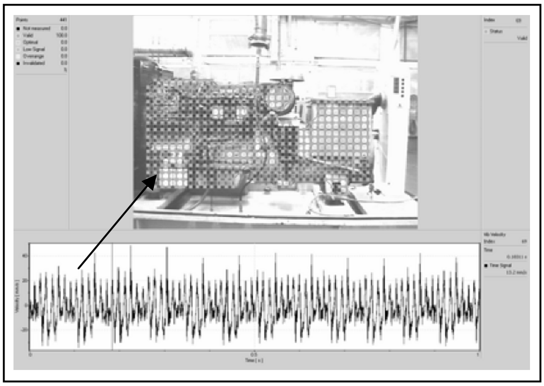

Figure 5. Typical vibration measurement for a point on the engine/alternator.

3.2 Acoustic Analysis

For the numerical simulation, all 700 vibration measurements were input to corresponding node locations on the engine/alternator surface mesh, Figure 6. The hard-linked fluid/structural mesh ensured that the driving vibrations would propagate into the fluid.

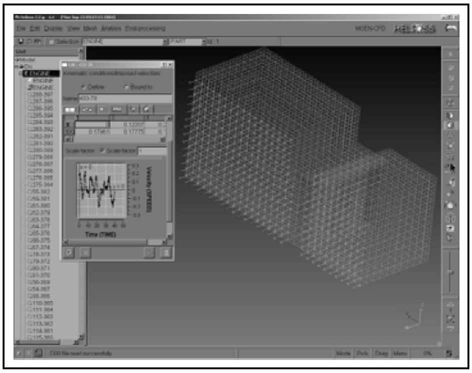

Figure 6. Measured surface vibrations imposed on node points on engine/alternator structural model.

The unsteady pressure fluctuations in the computational domain were computed utilizing the explicit time integration, including non-diffusive streamline upwind Petrov-Galerkin (SUPG) for momentum advection, and Large Eddy Simulation for turbulence. The model was run for a time period of approximately 20ms just sufficient to resolve the lowest frequency of interest, which in this case is 50Hz.

3.3 Acoustic Results

During the simulation the unsteady pressures and velocities were recorded on node points at the ventilation openings. A Fourier transform of these pressure time histories was completed and octave wide plots of Sound Pressure Level (SPL-dBA), calculated. Figure 7 shows an octave wide contour plot at a centre frequency of 105Hz. The SPL peak varies between 60dB(A) and 70dB(A) over the complete vent, Figure 7(a). For the case where the inlet vent was reduced in open area, identical to that of the airflow simulation, the SPL over all the open area is typically in excess of 70dBA, Figure 7(b). At a centre frequency of 210Hz, Figure 8, the SPL tends to vary in magnitude from approximately 85dB(A) to around 77dB(A) at the bottom on the complete inlet, Figure 8(a). For the reduced area inlet, Figure 8(b), the trend in SPL is markedly different to that of the full inlet, Figure 8(a). There is a band of SPL running through the top half of the vent between 80db(A) and 82dB(A). Further plots of SPL at a centre frequency of 1256Hz can be seen in Figure 9. Interestingly the trends and magnitudes of SPL are very similar for both vent configurations, Figure 9(a) and (b), and show evidence of higher acoustic modes. In practice these higher frequencies are attenuated by acoustic foam. Finally, average plots of SPL over the complete frequency range for both vent cases can be seen in Figures 10 (a) and (b). An increase of approximately 2dB(A) can be seen at the top of the reduced area vent, Figure 10 (b).

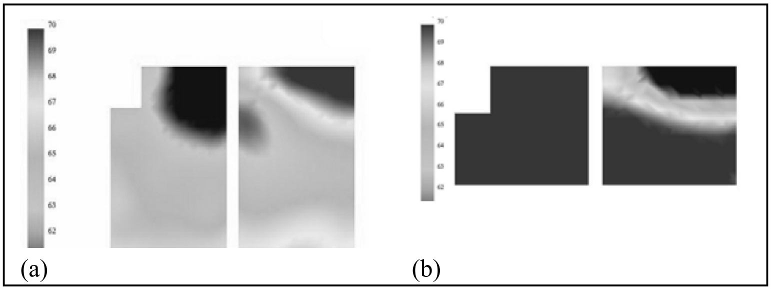

Figure 7. SPL dB(A) – CF 105Hz for (a) complete inlet and (b) reduced area inlet.

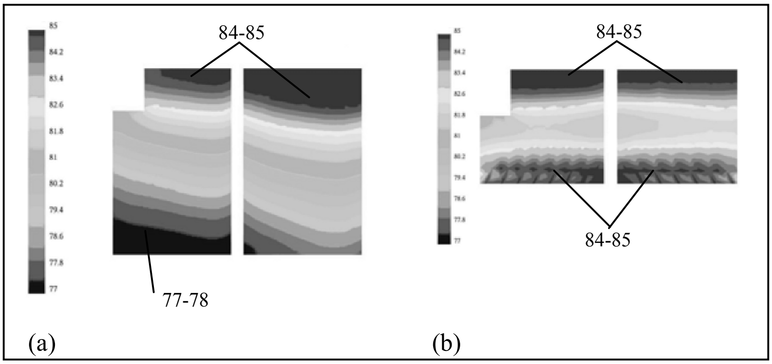

Figure 8. SPL dB(A) – CF 210Hz for (a) complete inlet vent and (b) reduced area inlet.

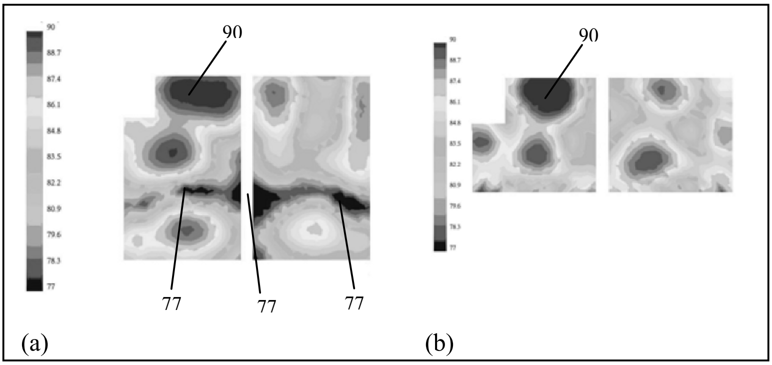

Figure 9. SPL dB(A) – CF 1265Hz for (a) complete inlet vent and (b) reduced area inlet.

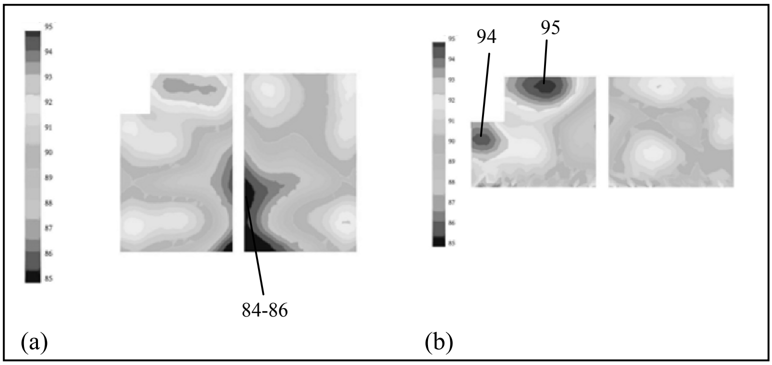

Figure 10. SPL dB(A) – Average for (a) complete inlet vent and (b) reduced area inlet.

4 Conclusions

For a typical enclosed engine generator set, virtual engineering techniques were employed to aid with the understanding of trade-off effects for a single parameter change with respect to cooling airflow and noise for a typical enclosed generator. The parameter change was induced by simply blocking off approximately one third of the ventilation open area.

The cooling airflow was modelled using CFD, and a multi-physics approach was employed to model the interaction between the vibrating engine/alternator and the surrounding fluid. Several hundred measured surface vibrations were directly input to the engine/alternator mesh model as a surface boundary condition to drive the acoustic propagation. SPLs were predicted over a frequency range of 50Hz to 2000Hz.

The overall airflow rate was unaffected for this particular parameter change, but the SPL magnitudes at the lower frequencies were increased by reducing the open area. Differences in acoustic duct modes, as a result of the parameter change, were visible.

Virtual engineering techniques can provide upfront design information pertaining to the trade-offs in question, giving insight into performance characteristics.

In this work, the airborne path only was considered due to computational constraints. Structural modes were not computed.

5 Acknowledgements

The authors would like to thank FG Wilson and Invest Northern Ireland for their grant contribution to the project.

6 References

[1] Mahon LLJ. Diesel Generator Handbook. Butterworth-Heinemann Ltd, 1992; 536-602.
[2] HD Ju, SB Lee, WB Jeong and BH Lee. Design of an acoustic enclosure with duct silencers for the heavy duty diesel engine generator set. Journal of Applied Acoustics 2004;65: 441-445.
[3] JW Choi, KE Kim and HJ Lee. Simple design method of the engine enclosure considering cooling and noise reduction. Journal of KSNVE 1999; 9(1): 184-8.
[4] Fluent 6.1 User’s Guide, Fluent Inc 2003-01-25.
[5] Mecalog, RADIOSS CFD user manual version 4.3.2001.
[6] Donea J. Arbitrary lagrangian eulerian finite element methods. Computational methods for transient analysis 1983;1:10.