A Computational Analysis of Flow and Acoustics around a Car Wing Mirror

Aidan J. Bowes, Reaz Hasan

Abstract— **The automotive industry is continually aiming to develop the aerodynamics of car bodies. This may be for a variety of beneficial reasons such as to increase speed or fuel efficiency by reducing drag. However, recently there has been a greater amount of focus on wind noise produced while driving.**

Designers in this industry seek a combination of both simplicity of approach and overall effectiveness. This combined with the growing availability of commercial CFD (Computational Fluid Dynamics) packages is likely to lead to an increase in the use of RANS (Reynolds Averaged Navier-Stokes) based CFD methods. This is due to these methods often being simpler than other CFD methods, having lower time and computing requirements.

In this investigation the effectiveness of turbulent flow and acoustic noise prediction using RANS based methods has been assessed for different wing mirror geometries. Three different RANS based models were used; standard k-ε, realizable k-ε and k-ω SST. The merits and limitations of these methods are then discussed, by comparing with both experimental and numerical results found in literature. In general, flow prediction is fairly comparable to more complex LES (Large Eddy Simulation) based methods; in particular the k-ω SST model. However acoustic noise prediction still leaves opportunities for more improvement using RANS based methods.

Index Terms— **Acoustics, aerodynamics, RANS models, turbulent flow**

I. Introduction

IT is widely known that desirable traits such as handling, customer comfort and car performance are affected by vehicle aerodynamics. Furthermore a good aerodynamic design will reduce overall drag which in turn will contribute greatly to lower emissions. This is very desirable due to the constant increase in fuel prices as well as the increasing demand by governments to meet environmental standards [1]. With the increase in popularity of electric cars in recent years [2], wind noise reduction has become an issue of greater importance; when engine noise is removed from a car all other noise sources become much more prominent factors [3]. This increase in exterior noise can have an effect on customer satisfaction levels.

Although traditionally analysis of these factors have always been done experimentally, usually by means of wind tunnel testing, recently the use of CFD has become more widely accepted as it can be a very powerful tool for analyzing these kinds of problems [4]. Flow around wing mirrors and the generated acoustics has been subject to investigation in numerous academic studies [5,6,7,8]. Moreover it is likely that there has also been further aerodynamic study performed on wing mirrors in both commercial car design as well as in vehicle design for motorsport. However due to the competitive nature of automotive development in these fields, most commonly motorsport, the results of these studies are usually not available for public access. This is due to the constant drive to produce more competitive cars with a key focus on body styling to reduce drag, which in turn leads to more radical aerodynamic designs. Design teams attempt to keep the performance of these designs secret for as long as possible in order to gain a competitive edge.

Of the numerous academic studies performed on generic wing mirror designs, the most popular numerical method used is LES although some hybrid methods such as DES (Detached Eddy Simulation) have been examined. This is likely due to the high level of accuracy that can be achieved from these types of analysis. However some of the most popular techniques employed in commercial aerodynamic analysis make use of RANS based methods as these involve relatively low computing resources compared to other methods. This can be an important factor to consider when the use of a slightly less accurate method may lead to a small decrease in the accuracy of the results obtained while offering a significant saving in computing requirements and consequently computing time and cost.

In this paper the commercial CFD program FLUENT 14.5 was used to predict transient flow behavior around two wing mirror shapes, one simple and one complex. The objective of this study was to assess the effectiveness of several popular RANS based CFD methods in predicting the flow around both a simple car wing mirror shape and also a more complex, realistic car wing mirror shape. The different solution methods as well as various setup parameters, e.g. meshing technique were chosen based on factors such as accessibility and computing restraints. This was done in order to properly demonstrate the level of accuracy it is possible to achieve when using the more accessible RANS based methods as compared to the often more accurate yet demanding LES based methods. Using accessible CFD methods to achieve accurate predictions of flow around complex shapes such as the wing mirror used in this study should demonstrate the ability for CFD to become a more accessible and widely used tool for a greater number of engineers.

Manuscript received January 29, 2015; revised February 20, 2015.

A.J. Bowes was a BEng(H) student in the Dept. of Mechanical and Construction Engineering, Northumbria University, Newcastle Upon Tyne, NE1 8ST, UK

R. Hasan is with the Dept. of Mechanical and Construction Engineering, Northumbria University, Newcastle Upon Tyne, NE1 8ST, UK (e-mail: Reaz.Hasan@northumbria.ac.uk).

ISBN: 978-988-14047-2-5
ISSN: 2078-0958 (Print); ISSN: 2078-0966 (Online)

II. PROCEDURE

A. Domain and Boundary Specification

All calculations were carried out using the commercial CFD package ANSYS FLUENT 14.5 software. The methodology involves the iterative solution of the Navier-Stokes equations with the finite volume method on an unstructured mesh configuration. For more information see Versteeg et al [9]. Three different RANS based models were used in order to provide a reasonable comparison of effectiveness; standard k-ε, realizable k-ε and k-ω SST.

In order to validate the test procedure for the 3D wing mirror cases it was first necessary to validate the accuracy of the models that were to be used. This was done by performing analysis on a 2D cylinder to find key flow features such as drag coefficient, drag force and Strouhal number. Once the 2D method had been established as effective, analysis was carried out on the 3D geometries.

2D model dimensions were chosen based on a variety of factors. Firstly it was necessary to relate the geometry to a real wing mirror as closely as possible. Therefore a diameter of 0.2m was chosen for the cylinder as this is a representative dimension of a real wing mirror. Additionally, a distance of 1.2m from the cylinder center to the top boundary was chosen in order to represent the distance from a wing mirror to the ground. Other dimensions were then applied based on boundaries in similar tests from literature [5]. A fluid velocity of 40m/s was used as this is an accurate representative speed for a road vehicle. Fig.1 below shows the 3D boundary for the simple geometry with dimensions given in terms of the cylinder diameter D.

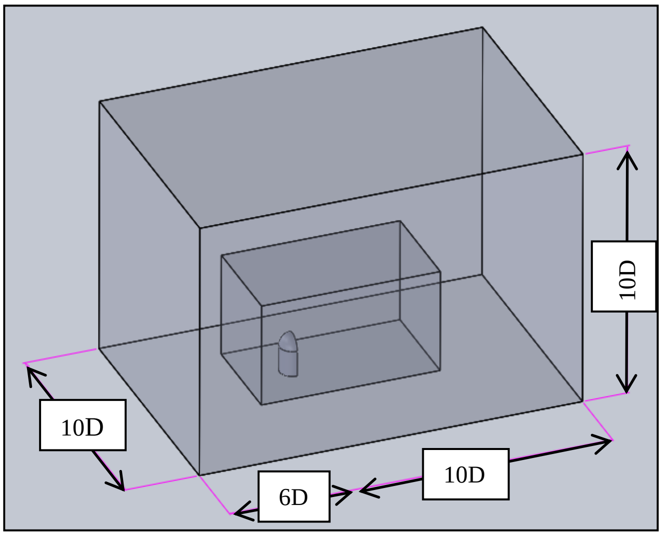

Fig.1: Simple geometry and boundary dimensions

B. Grid Generation and Time Step Selection

In order to capture certain oscillating flow features it was necessary to select an adequate time step size. Here the equation St=fl/v was used to find the approximated frequency where St = Strouhal number, f = frequency, l = characteristic length and v = velocity. This frequency could then be used to find an approximate time period. It is recommended that transient cases such as this should be solved by at least 30 time steps per cycle [11]. Using these values, a recommended time step of 6.67×10^{-4} was calculated for the computation. A time step of 5×10^{-4} was selected as this was considered to be a simpler number to work with; as this is a smaller time step, accuracy would only be improved by this change. After these parameters had been established it was possible to run the 2D simulations for each of the three models. The values found were then compared to values found using an analytical method for verification. The two 3D wing mirror geometries were created using the SolidWorks CAD package. The simple geometry was created in order to be comparable with the generic wing mirror found in other literature [5,6,7]. It is comprised of half a cylinder with a diameter and length of 0.2m blunted by quarter of a sphere of the same radius. The complex wing mirror geometry was designed to be a more realistic representation of a wing mirror. Dimensions were taken using a Vauxhall Zafira wing mirror as a base. Both geometries were modelled mounted on a flat plate. Boundary conditions as well as fluid velocity were modelled taking into consideration realistic driving conditions. The same boundary conditions including inlet velocity and time step were maintained for each geometry, with the mirror having an identical positioning on the plate. Fig.2 and Fig.3 show in more detail the simple 'generic' geometry and the complex geometry respectively.

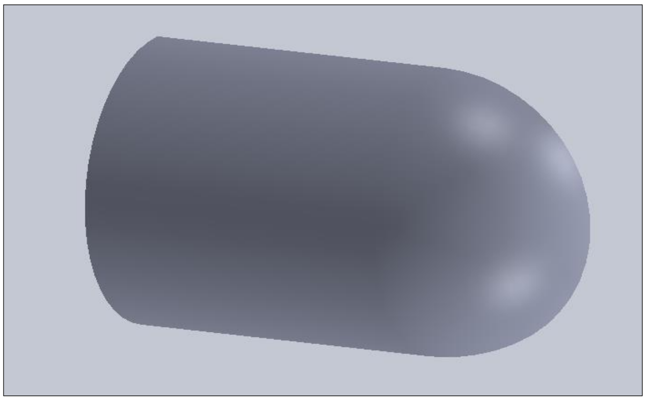

Fig.2: Simple 'generic' wing mirror geometry

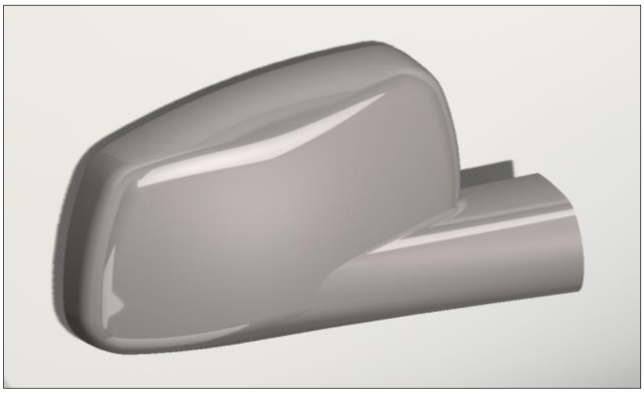

Fig.3: Complex wing mirror geometry

An unstructured meshing technique was used in ANSYS in order to create a mesh for both 3D geometry cases. This was chosen as it was the most accessible meshing technique taking into consideration the project time frame as well as expertise in mesh creation and mesh size limitations imposed by the available software licenses. Each mesh included local

ISBN: 978-988-14047-2-5
ISSN: 2078-0958 (Print); ISSN: 2078-0966 (Online)

refinement techniques in the way of bodies of influence and face sizing controls in order to limit element sizes in the areas where the flow is likely to be most complex such as near walls and immediately downstream of the wing mirror [11]. These features were included to increase accuracy in these areas. Also layered prismatic elements were incorporated at all wall boundaries as they are known to have good alignment with flow in these areas, effectively helping to capture the boundary layer more accurately.

Y+ values for both geometries were found to be within the range suggested by FLUENT best practice [11]. Fig.4 below shows cross sections of the simple geometry mesh, showing clearly how bodies of influence have been applied during the meshing process.

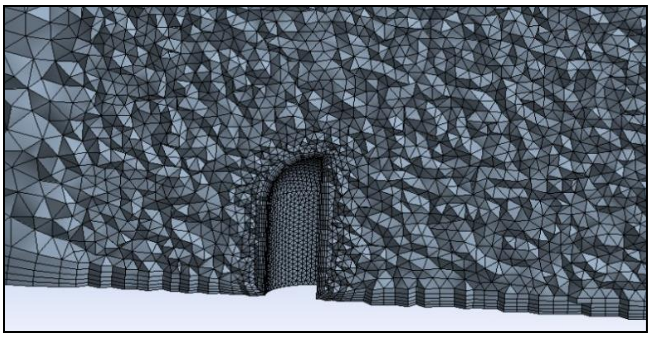

Fig.4: Close-up mesh cross section

C. Acoustic Formulation

Using the 3D test method transient simulations were run until the solution reached a steady state. Once this had been achieved the FWH (Ffowcs-Williams and Hawkings) model was then switched on in FLUENT and acoustic receiver locations, plus acoustic sources were specified. The FWH model is fundamentally an inhomogeneous wave equation derived from the continuity equation and the Navier-Stokes equations containing monopole, dipole and quadrapole source terms. For a stationary surface the monopole term vanishes. When Mach number is less than 0.2 the contribution of quadrapole is negligible. Leaving the dipole term which can be expressed in terms of PFL (pressure frequency levels) as

$$\mathrm{PFL}(y) = 20\log_{10}(p_{sf}(y,t)_{rms}/p_0) \quad (1)$$

Where y is the surface location where the pressure fluctuations are monitored [11]. The solution was then allowed to continue for a number of cycles. Once this further iteration had finished acoustic pressure signals could be post-processed using fast Fourier transform capabilities in FLUENT and data pertaining to the flow characteristics could be analyzed.

In setting up the FWH acoustic model it was important to consider receiver locations, due to computational restraints there was a limit to the amount of acoustic receiver locations that could be defined in order to maintain a practical computing time for the model. Therefore it was decided that 12 receiver locations would be defined. These would be located at points on the wing mirror and the adjoined surface where it was suspected that acoustic pressure fluctuations would be high, based on literature [6,7]. This comprised of three points along the rear surface of the mirror and 9 points in the wake of the mirror on the mounting surface. A diagram of these locations for the simple geometry can be seen in Fig.5 with points labelled 1-12 for reference.

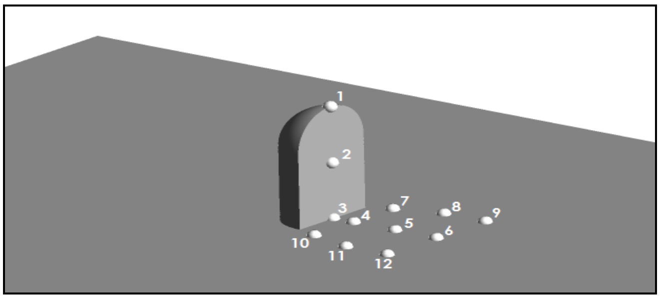

Fig.5: Acoustic Receiver Points

III. Results

A. Flow

For each of the three models used the flow separation point has been examined. Fig.6 and Fig.7 show contours of wall shear stress on the wing and mounting surface for both the simple and complex geometries respectively using the k-ω SST model.

It can be observed that for all of the models used it was not possible to predict a flow separation upstream of the mirror edge for the simple geometry. Instead all of the models seem to predict the same trend that after the flow has come to rest from the initial collision with the mirror it then begins to accelerate due to the rapidly curving geometry. This velocity reaches a peak in an area approximately 0.15D upstream of the mirror edge, causing the high wall shear stress in this area (Fig.6). This is where LES as well as experimental methods predict the boundary layer separation [6]. However the RANS methods seem to predict that due to the now slower curving geometry, from this point flow decelerates before separating at the mirror edge. It is interesting to note that although separation is not predicted, the point of highest wall shear is very similar to the experimental separation point. The flow then reattaches approximately 3D downstream of the mirror, this agrees with literature [5,6].

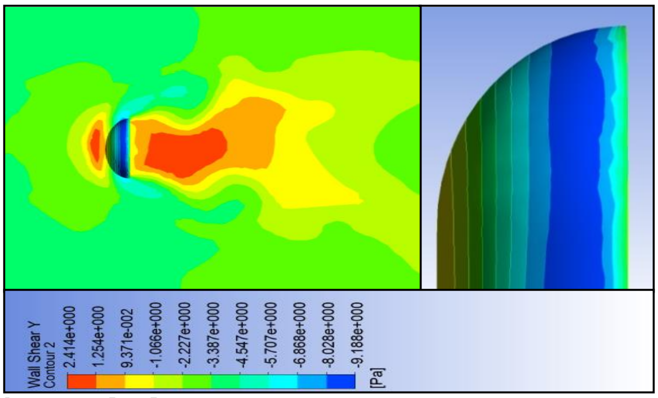

Fig.6: Contours of wall shear stress for simple geometry

In contrast to this, the k-ω SST model predicts a slightly different flow separation for the complex geometry. Although it shows the majority of the flow to separate from the wing in the center of the edge radius, it is clear that some flow separation is happening upstream of this point at the

ISBN: 978-988-14047-2-5
ISSN: 2078-0958 (Print); ISSN: 2078-0966 (Online)

wing mirror tip. The areas where the flow separation is predicted are areas where a significantly large amount of fast flowing air will be concentrated.

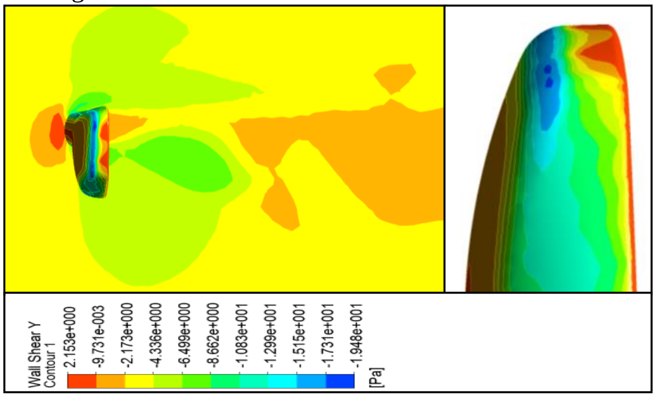

Fig.7: Contours of wall shear stress for complex geometry

Despite some discrepancies in predicting the separation point, the predicted characteristic vortex formation corresponds with literature [5]. All models clearly show the presence of one large vortex region extending for approximately 3D in the wake of the cylinder. Moreover all models show the presence of a 'horseshoe vortex' located on the mounting plate upstream of the wing mirror body. Fig.8 shows the Q-criterion for the k-ω SST model for both geometries; this is a scalar used to visualize vortex formation and represents the local balance between rotation and strain rate. It is sometimes preferred as a visual representation of vortex formation over variables such as contours of vorticity, or the pressure field and is often included for visualization purposes in relevant literature [5,6]. This has been colored by velocity magnitude in order to provide an idea of velocity in turbulent areas.

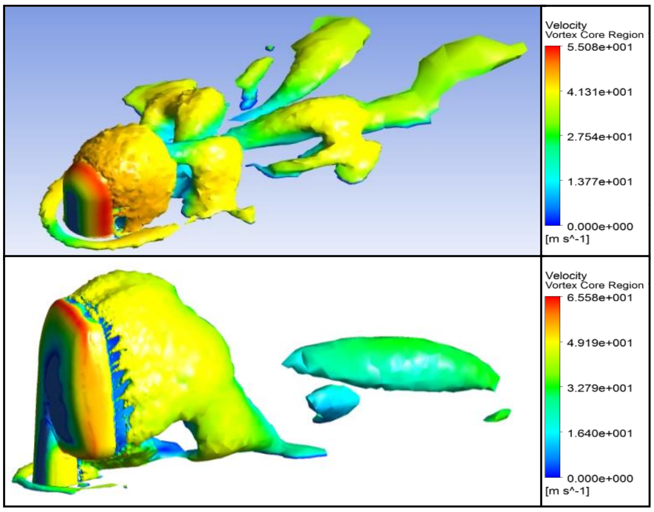

Fig.8: Q-criterion for both the simple and complex geometries

It can be seen that the turbulent structures do not continue as far in the wake of the mirror for the complex geometry. This may be because whereas the simple geometry connects directly to the mounting plate causing a large projected diameter at the base, the complex mirror incorporates a type of 'stem', allowing some air to flow through the gap between the mirror and the plate. This in turn reduces the vortex region in the mirror wake and the vortex interaction with the plate, which should act to reduce the propagation on turbulent structures further downstream. Fig.8 also shows the area of high velocity at the sharp leading radius where it is expected flow separation may occur.

It is worth noting that the Q-criterion plot for all of the RANS models used is not comparable to those produced using LES techniques in literature [5,6]. This is due to RANS models being unable to show the characteristic 'large eddies' that would be produced. However, when comparing visualizations of velocity vectors to those of Belamri et al [5], it can be seen that flow behavior is very similar in the k-ω SST model. Fig.9 shows vortex formation in the wake of the simple mirror in the form of velocity vectors for the k-ω SST model. It is suggested that the vortex interaction with the mounting surface approximately 1-2D downstream of the mirror edge is one of the major flow induced noise sources. Fig.6 shows this interaction well.

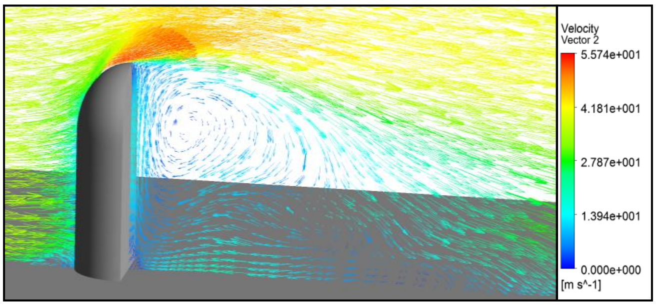

Fig.9: Velocity vectors predicted by k-ω SST model

It was found that vortex shedding from the curved edge of the complex geometry is minimal when compared to that for the sharp edge of the simple geometry, reducing vortex interaction with the plate. This is interesting when compared to the simple geometry and may act to reduce the acoustic noise created greatly as vortex production from the mirror edge has been identified as a major source of noise [7].

B. Acoustics

Acoustic data obtained through the FFT of recorded sound pressure levels at points on the wing mirror will be presented. The presented results will be focusing primarily on two points. One at the tip of the wing mirror, and one approximately 1.5D downstream of the mirror edge, corresponding to points 1 and 12 (Fig.5). This is where the greatest sound pressure levels are expected to originate from based on the vortex interaction with the mirror and the plate

Fig.10 shows acoustic results for the simple geometry at points 1 and 12 for the three different models. An observation is that the frequency range is only from 0-100Hz, which is relatively small compared to acoustic data that can be seen in literature [5,6,7]. This may be because unlike in LES methods, RANS methods cannot predict the vortex formation to a sufficient accuracy for the many vortices that may be created and shed at higher frequencies, or that may be very small and close to wall areas. Instead, one large vortex region is shown which is likely to vary at a lower frequency. Another comparison to literature is that RANS methods seem to under-predict the SPL (Sound Pressure Levels) when compared to a LES or experimental method. This was also found by Ask et al [7] when using a DES method which has RANS like behavior near wall.

Therefore this inaccuracy may also be due to the RANS method dampening turbulent pressure fluctuations due to a disproportionate production of turbulent viscosity. This would also explain the poor prediction in boundary layer separation.

In comparing the three RANS methods it is clear for the simple geometry the two k-ε based models seem to predict the same trend showing only two significant peaks. Only one of these peaks corresponds to a sound greater than conversational level. However the k-ω SST model predicts on average a much higher sound pressure level at lower frequencies with multiple peaks at approximately 15Hz, 25Hz and 40Hz. The trend exhibited here is not unlike that shown in literature for the DES model [7] albeit at a lower SPL, which is to be expected. All of these peaks are above an average conversational level and would present themselves as very loud, low sounds to the observer which may be a cause for noise complaints from observers and the driver, with values approaching 100dB being produced from the tip of the mirror.

Fig.11 shows the predicted SPL predicted by the k-ω SST for the complex geometry. This is still predicting variation in sound pressure across the frequency range yet with a reduction in higher frequency noises, possibly due to the reduction in vortex interaction with the plate.

Overall based on the comparison of the results it can be assumed that of the RANS methods used the k-ω SST model is the most accurate based on the greater accuracy in predicting flow characteristics. However due to the nature of RANS models, all predicted pressure levels are dampened when compared to those found by a LES method. Conversely, even with a lower SPL prediction, very loud noise levels were detected at a lower frequency range which would already cause disturbance, a more accurate prediction would likely increase these noise levels.

Fig.12 is included to compare areas of high sound pressure levels for both geometries using the k-ω SST model. This shows root mean squared pressure change with respect to time. Areas of highest pressure change are expected to be where there is most vortex interaction, which will in turn be where the highest SPL will be recorded. Here it is immediately apparent why the simple geometry shows more variation in SPL with many more areas exhibiting a high pressure change. This is due to vortex shedding from the sharp edge of the mirror.

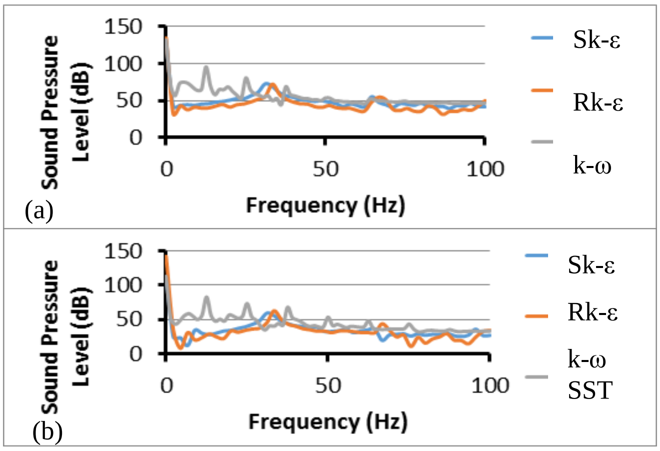

Fig.10: SPL vs frequency for simple geometry at (a) point1 (b) point12

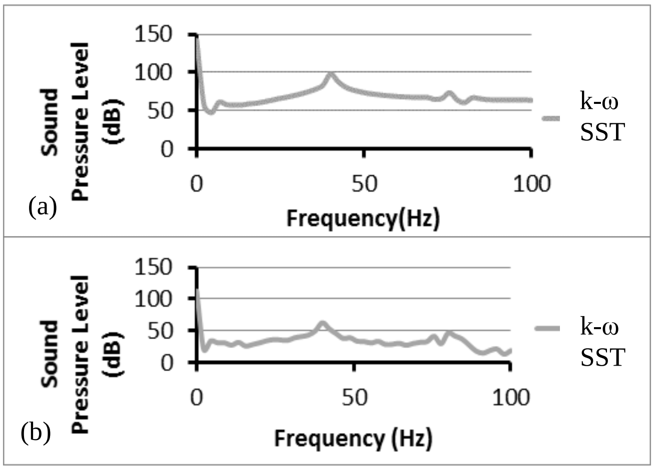

Fig.11: SPL vs frequency for complex geometry recorded at (a) point1 (b) point 12

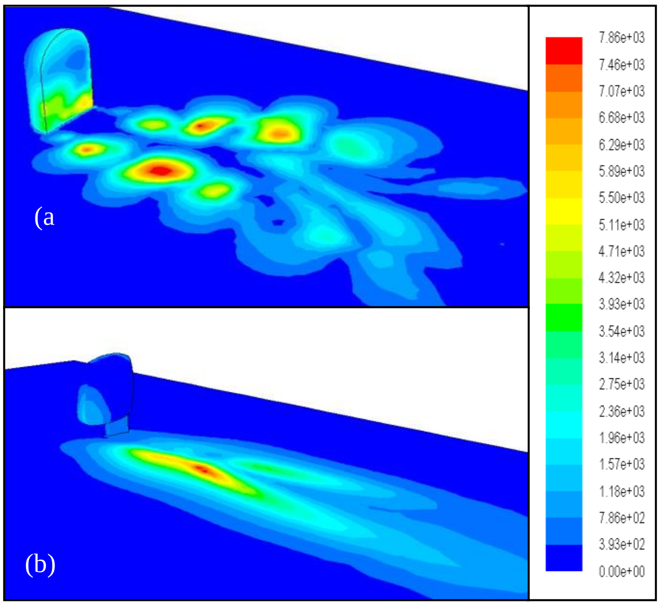

Fig.12: Contours of rms dpdt for (a) simple geometry (b) complex geometry

IV. Conclusion

Analysis was initially performed on a simple 2D geometry using three different turbulence models; standard k-ε, realizable k-ε and k-ω SST. After satisfactory results were obtained analysis was carried out on two different 3D geometries using the same three turbulence models, one simple and one complex.

It has been found that the RANS models used are able to predict turbulent flow features with some accuracy; especially the k-ω SST model which has demonstrated the highest performance throughout. This can be seen in the prediction of more complex turbulent vortices, which may possibly be comparable to those shown in a LES method, as well as the better prediction of flow separation, although this is still not comparable to what can be found by a LES method.

Acoustic post-processing was then performed with k-ω SST showing some promise in SPL prediction especially compared to the excessive under-prediction of the k-ε based methods. However, overall acoustic results left something to be desired. This was due to the nature of the RANS methods

ISBN: 978-988-14047-2-5
ISSN: 2078-0958 (Print); ISSN: 2078-0966 (Online)

used and was to be expected as even in cases where RANS methods have been used to obtain a reasonable acoustic solution interest has been expressed in using a LES based method as this is likely to be more accurate [12]. Conversely it could be argued that the level of accuracy obtained using this acoustic method may be enough for the average user, more so at an entry level to CFD. Furthermore this study may help in providing enough information when attempting to make the decision between required computing cost and accuracy.

REFERENCES

[1] Hucho, W.H., 1987, Aerodynamics of Road Vehicles, Butterworths, London, pp. 59-67, Chap. 2

[2] The Guardian website (Accessed 31/10/13) http://www.theguardian.com/environment/2013/jan/08/electric-car-sales-2012

[3] Otto, N., Simpson, R., Wiederhold, J., 1999, "Electric Vehicle Sound Quality",1999-01-1694, SAE Technical Paper 1999-01-1694, SAE International

[4] Massey, B.S., 2012, Mechanics of Fluids 9th Edition, Spon Press, Abingdon, pp. 353

[5] Belamri, T., Egorov, Y., Menter, F. R., 2007. “CFD simulation of the aeroacoustic noise generated by a generic side view car mirror.” In Proceedings of 13th AIAA/CEAS Aeroacoustics Conference, AIAA-2007-3568.

[6] Afgan, I., Moulinec, C., & Laurence, D., 2008. “Numerical simulation of generic side mirror of a car using large eddy simulation with polyhedral meshes.” International journal for numerical methods in fluids, 56(8), 1107-1113.

[7] Ask, J., & Davidson, L., 2006. “The sub-critical flow past a generic side mirror and its impact on sound generation and propagation.” AIAA Paper, 2558, 2006

[8] Yang, Z., Gu, Z., Tu, J., Dong, G., & Wang, Y., 2014. Numerical analysis and passive control of a car side window buffeting noise based on Scale-Adaptive Simulation. Applied Acoustics, 79, 23-34.

[9] Versteeg, H. K., & Malalasekera, W. 2007. An introduction to computational fluid dynamics: the finite volume method. Pearson Education.

[10] Lanfrit, M. 2005. “Best practice guidelines for handling automotive external aerodynamics with FLUENT.” ANSYS FLUENT Paper

[11] Ffowcs, W.J., Hawkings, D., “Sound generation by turbulence and surfaces in arbitrary motion.” Philosophical Transactions of the Royal Society 1969 A264: 321-342

[12] Algermissen, G., Siergert, R., Spindler, T., 2001, “Numerical Simulation Of Noise Generated By Fans Under Installation Conditions”, 7th AIAA/CEAS Aeroacoustics Conference and Exhibit, AIAA, Maastricht

ISBN: 978-988-14047-2-5
ISSN: 2078-0958 (Print); ISSN: 2078-0966 (Online)