Validation of Different Fan Modelling Techniques in Computational Fluid Dynamics

Randi Franzke\textsuperscript{1,2} and Simone Sebben\textsuperscript{1}

\textsuperscript{1}Department of Mechanics and Maritime Systems
Chalmers University of Technology, Gothenburg, Sweden

\textsuperscript{2}Volvo Car Corporation
Gothenburg, Sweden

Abstract
The accuracy of predicting the engine bay flow field with computational fluid dynamics (CFD) is crucial for designing efficient cooling systems for heat sensitive components. The engine cooling fan is the main driving component in cases of high thermal load, such as uphill driving with a trailer, or high speed driving on a highway, when the ram air itself is no longer sufficient for cooling purposes.

The most widely used fan modelling method is the Moving Reference Frame (MRF). This method can be used in steady and unsteady simulations, but has the drawback of using a fan geometry that is fixed in the global reference frame and, therefore, causing non-physical low velocity regions in the wake of the blades. The Rigid Body Motion (RBM or “sliding mesh”) approach is a more accurate, but also more expensive approach, since it uses an unsteady solver. This study looks closely at the prediction of the flow field in the wake of an axial fan for different freestream velocities and fan speeds using the traditional MRF and RBM approach. In addition, a method that uses the average of flow field data for multiple MRF simulations with different fan positions is presented. Thereby the shadow of the fan blades is removed from the wake and the flow field becomes more uniform without the need of performing unsteady simulations. As a reference, measurements are performed on a vehicle fan with a 2D Laser Doppler Anemometry set-up in a small scale wind tunnel.

The results show good agreement between the measurements and the RBM simulations. As expected, the MRF simulations show a distinct blade pattern in the wake flow field. This was successfully removed by the proposed averaged MRF method. Even though there are still some differences between this method and the experimental results, the average MRF method has shown to be applicable as it improves the flow field results at a relatively low computational cost.

Introduction
Electric vehicles (EV) are currently the most promising solution for reducing CO\textsubscript{2} emissions in individual transportation. However, EVs contain more components that are sensitive to high heat inside the engine bay than a traditional vehicle with an internal combustion engine. Therefore more accurate tools to predict the flow field in the wake of an axial fan for different freestream velocities and fan speeds using the traditional MRF and RBM approach. In addition, a method that uses the average of flow field data for multiple MRF simulations with different fan positions is presented. Thereby the shadow of the fan blades is removed from the wake and the flow field becomes more uniform without the need of performing unsteady simulations. As a reference, measurements are performed on a vehicle fan with a 2D Laser Doppler Anemometry set-up in a small scale wind tunnel.

The most widely used fan modelling method is the Moving Reference Frame (MRF). This method can be used in steady and unsteady simulations, but has the drawback of using a fan geometry that is fixed in the global reference frame and, therefore, causing non-physical low velocity regions in the wake of the blades. The Rigid Body Motion (RBM or “sliding mesh”) approach is a more accurate, but also more expensive approach, since it uses an unsteady solver. This study looks closely at the prediction of the flow field in the wake of an axial fan for different freestream velocities and fan speeds using the traditional MRF and RBM approach. In addition, a method that uses the average of flow field data for multiple MRF simulations with different fan positions is presented. Thereby the shadow of the fan blades is removed from the wake and the flow field becomes more uniform without the need of performing unsteady simulations. As a reference, measurements are performed on a vehicle fan with a 2D Laser Doppler Anemometry set-up in a small scale wind tunnel.

The results show good agreement between the measurements and the RBM simulations. As expected, the MRF simulations show a distinct blade pattern in the wake flow field. This was successfully removed by the proposed averaged MRF method. Even though there are still some differences between this method and the experimental results, the average MRF method has shown to be applicable as it improves the flow field results at a relatively low computational cost.

In this paper, the flow field resulting from the MRF and RBM model are compared with each other and with experimentally obtained data. Furthermore, another approach is presented that averages the flow field information over multiple MRF simulations and is, therefore, expected to improve the simulation results without adding considerable computational effort.

Experimental Set-up
The Model Wind Tunnel (MWT) used for the experiments is a 1/5\textsuperscript{th} scale version of the full-scale wind tunnel at the Volvo Car Corporation. The tunnel is of a closed-loop, slotted-wall type with a cross-section of 1.32 x 0.82 m\textsuperscript{2}. For this study, however, the slots were taped over, creating a closed test section. This was done in order to facilitate the comparison with the simulation data. The test object is a Volvo V40 production fan with a diameter of 0.38 m, in its shroud. The fan is placed in the test section so that the fan rotation axis is in the centre of the wind tunnel cross section. Due to the asymmetry of the shroud, the space to the left and right of the shroud is not equally large. The set-up can be seen in Figure 1, with the LDA probe mounted on a yz-traverse downstream of the fan.

The investigated rotation speeds are 1400 rpm and 2800 rpm at a ram air flow of 5 m/s. The measurements are performed with a Laser Doppler Anemometry system by Dantec Dynamics. The probe is a 2D probe, with a focal length of 50 mm. In the setup the probe is placed perpendicular to the flow to measure the u and w component of the flow in a yz-plane x = 0.04 m downstream of the fan’s motor. The seeding is done with a battery
driven and remote controlled smoke machine, using a fog fluid on water-glycol basis. The fog generator is located downstream of the fan, to provide a more even seed particle distribution.

**Numerical Set-up**

All simulations are performed with StarCCM+ (Ver. 11.06) from Siemens. The computational domain has the same cross section as the model wind tunnel. However, it is a bit longer than the test section, allowing the flow to settle. The mesh consists of $8 \times 10^6$ polyhedral cells inside the fan domain and $3 \times 10^6$ polyhedral cells in the remaining air domain. This yields a total of $13 \times 10^6$ cells, and a $y^+$ $<$ 1 at the fan blades. A mesh dependency study was performed in which the area downstream of the fan was further refined, resulting in a total of $23 \times 10^6$ cells. No significant changes in the flow field and the velocity contours could be observed.

Three different methods are used to model the fan rotation. First, the Moving Reference Frame (MRF) method is used. This model introduces a relative rotation to the respective user-defined region, while the fan blade geometry is fixed in the global reference frame. This model is commonly used, since it allows inexpensive steady-state simulations of rotation and/or translation. Second, the Rigid Body Motion (RBM) is studied. In this approach, the whole fan region is moving at the specified rotational speed. It is commonly recommended to use a time-step that represents a turn of less than 1 deg. Therefore, it is a very time consuming approach, but expected to give good results. The third method tested, is an averaged MRF approach. Since the fan is symmetric and has 8 blades, the geometry is repeating itself every 45 deg. Hence it is sufficient to cover this range by 9 simulations where the blade positions are changed by increments of 5 deg. A parametric study is conducted where the adaptation of this model to increments of 45 x 1 deg and 3 x 15 deg were tested. Starting from a converged standard MRF simulation, each new fan position required only 500 additional iterations before convergence was reached. The solution was considered to be converged, when all residuals of the governing equations decreased by two orders of magnitude and the maximum velocity stabilised.

The simulations based on the MRF methods were performed using a Reynolds-Averaged Navier Stokes (RANS) solver, using the realisable $k$-$\varepsilon$ turbulence model. The RBM simulations were performed with an unsteady RANS (URANS) solver, also using the realisable $k$-$\varepsilon$ model. For the comparison of RANS and URANS data, the URANS data is time-averaged over 10 fan revolutions.

**Results**

In this section, the simulation results for all three methods are compared to measurement data that was taken 0.04 m downstream of the fan. The freestream velocity was 5 m/s, the fan speeds were 1400 rpm and 2800 rpm, respectively. The comparison is done in two parts. First, a comparison is done by looking at the axial velocity over the complete wake (marked red in Figure 2). Here, the measurements were taken in steps of 20 mm in the $yz$-plane. Second, the velocities for the different simulation methods are plotted along two vertical lines (marked blue). Here, the velocities were measured every 5 mm in z-direction. The uncertainty in the measurement of the velocity component in axial direction is less than 0.3 m/s.

The flow field obtained by the MRF method shows the largest differences from the three simulation models compared to the measurements. The main difference is, that the flow field is disturbed by the fixed blade position, which leads to an uneven distribution. Averaging the flow field for 9 different fan positions with the average MRF approach gives a more uniform
flow field, which is comparable to the RBM simulation and the measurements.

However, there are two major differences between the MRF based simulations and the RBM simulation/measurement results. The first is an area of relatively low velocities on the top of the wake, right of the control unit. In the RBM simulation and the measurements, this area shows significantly higher axial velocities. An unsteady MRF simulation was performed to determine if this can be traced back to differences between the RANS and the URANS model. Since the URANS MRF showed the same behaviour, it is concluded, that this low velocity region is resulting from an interaction between the MRF interface and the geometry of the shroud. The reason why the other struts do not show the same effect on the flow field is, that the strut holding the control unit on top has only a distance of 4 mm to the MRF interface, while the other struts are 10 mm downstream of the interface. The second difference that is observed, is an elevated velocity behind the struts in direction of rotation for the two MRF based simulation approaches. For the RBM simulation it is hardly visible in Figure 3, but also here there are small areas with slightly elevated axial velocities. However, those areas are on both sides of the struts and of approximately equal size and magnitude. The resolution of the measurement grid is insufficient for these kinds of observations.

Figure 4 shows the same wake plane for the rotation rate of 2800 rpm. Due to the higher rotation rate, the scale had to be adjusted, but the same observations as for the 1400 rpm case can be made.

![Figure 4: Axial velocity field for a rotation rate of 2800 rpm and a freestream velocity of 5 m/s](image)

**Vertical Lines**

For a closer comparison, the axial velocities along two vertical lines were plotted, one at the centerline (Figure 5) and one with a 0.1 m offset in positive y-direction (Figure 6). The measurements were repeated with a step size of 5 mm in z-direction. To facilitate the reading of the graphs, the cross-section of the fan region and the shroud is added to the plots.

Figure 5 shows the measured and simulated axial velocities along the centerline of the fan, for 1400 (left) and 2800 rpm (right). Overall, all three fan modelling approaches match the experimental data relatively well for the lower rotation rate. The largest differences occur in the upper half \((z > 120 \text{ mm})\), close to the control unit. There, lower axial velocities have been observed and described in the previous section (compare Fig. 3 and 4). For the 2800 rpm case, just above the hub (90 < z < 120 mm), the measured velocities are considerably lower than the ones obtained from the CFD simulations. Since this result was repeatable, a possible reason might be that the seeding did not reach the area. No notable difference between the MRF and the averaged MRF results can be seen along the centerline.

![Figure 5: Axial velocity at 5 m/s for 1400 rpm (left) and 2800 rpm (right) 0.04 m behind the fan on the centerline (comp. Fig. 2)](image)

Figure 6 shows the axial velocity over a vertical line that is located 0.1 m to the right of the center of rotation, as seen from the downstream view. For the 1400 rpm case (left), all simulation models agree well with the measurement data. It can be observed, that the wake for the MRF based simulations has a smaller extent in z-direction than the wake in the RBM results or the measurement data. At 2800 rpm large differences between the standard (single) MRF and averaged MRF method can be seen on both sides of the hub. The velocities for the standard MRF model are fluctuating notably. The averaging over multiple MRF simulations with different fan positions successfully removes these fluctuations and achieves a uniform flow field distribution similar to the RBM and measurement data.

![Figure 6: Axial velocity at 5 m/s for 1400 rpm (left) and 2800 rpm (right) 0.04 m behind the fan 0.1 m to the right of the centerline](image)
Average MRF - Increment Study

Since the first attempt of averaging the flow field obtained by multiple MRF simulations showed a notable improvement weighted with a low additional computational effort, it was decided to investigate the impact of the step size in rotation on the results. For this particular fan, the blade geometry repeats itself every 45 deg, therefore it is only necessary to cover this range. In addition to the previously presented 9 positions with a 5 deg angle increment, cases with finer (45 x 1 deg) and coarser (3 x 15 deg) increment are examined.

Figure 7 shows the wake plane 0.04 m downstream of the fan where the 45 deg range is covered by increments of 1 deg, 5 deg and 15 deg at a rotation rate of 2800 rpm. There are no visible differences between the resulting flow fields obtained by using the average over 45 MRF simulations with each 1 deg difference in fan blade position or 9 simulations with an increment of 5 deg. Similar results were also obtained when averaging over 3 x 15 deg positions. However, for this case, some small blade shadows start to re-appear.

Figure 8 shows a comparison for the same increments along the two vertical lines that were presented in the previous section. Here, an increase or decrease of averaged MRF simulations does not show any notable effect.

Conclusions

In this paper, results comparing three different fan modelling approaches with measurement data, obtained using a 2D LDA system, are presented. The commonly used standard Moving Reference Frame (MRF) model showed large differences to the experimental data, since the fixed blades caused an imprint in the wake and therefore an uneven flow distribution. It was shown that it is possible to cost-efficiently remove those imprints by averaging the flow field over multiple steady-state MRF simulations with different fan positions. A parametric study demonstrated that the simulation of two additional fan positions was already sufficient in the presented test set-up to substantially improve the resulting flow field. Although the Rigid Body Motion model with URANS simulation showed the best agreement with the measurement data, it also requires the largest computational effort. Hence, the proposed average MRF approach is a good alternative to the common fan modelling methods, as it gives improved results with minimal increased computational costs.

Acknowledgements

The authors would like to thank the Volvo Car Corporation for their support in this study. We would like to especially thank David Englen for his support in the experimental work, as well as Tore Bark for many interesting discussions of the results. Furthermore, we thank the Swedish Energy Agency for the funding of this project, as well as the Swedish National Infrastructure for Computation (SNIC) for providing computational resources.

References


