

Automotive Axial Fan

The steady-state rotating reference frame model in FLUENT has been used by Siemens VDO Automotive to study the performance of an axial fan used for automotive applications. The static pressure rise for the AMCA chamber test conditions was computed, and was found to be in good agreement with the experimentally measured data. In addition, the CFD analysis provided a wealth of data that would be hard to obtain otherwise.

A six-bladed axial fan with a rotating shroud has been analyzed using FLUENT. The fan has a small hub to tip ratio (0.44), with a nominal blade diameter of 290 mm. The tips of the fan blades are attached to the shroud, which is L-shaped and has an outer diameter of 312 mm. The shroud overlaps a flat plate that is part of the fan housing. A 3 mm gap separates this plate from the shroud. The fan rotational speed is 2500 rpm, which, in air, corresponds to a Reynolds number (based on the blade chord length) of 130,000.

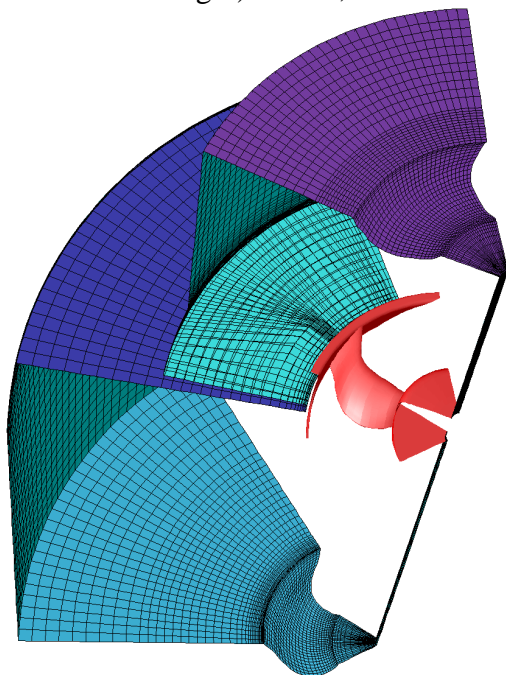


Figure 1: The geometry and grid of the one-sixth model of the fan and housing

Taking advantage of the periodicity in both the geometry and the flow, it is sufficient to model only one-sixth of the actual flow environment. For this reason, only a single passage-to-passage volume was analyzed (with the blade at the center). The flow equations were solved in the reference frame of the rotating fan, thus allowing for a steady-state treatment of the problem. The standard k- ϵ turbulence model was used.

Figure 1 shows the geometry and mesh of the one-sixth model of the axial fan. Air enters through the mass flow inlet at the top (purple), passes through the fan, and exits through a pressure outlet at the bottom (light blue). Enclosed in the fan housing is the fan blade with the attached rotating shroud and hub (red). Periodic boundaries are used on the sides of the domain, and allow the passage of fluid from one sector to the next. These have been removed in Figure 1 for image clarity.

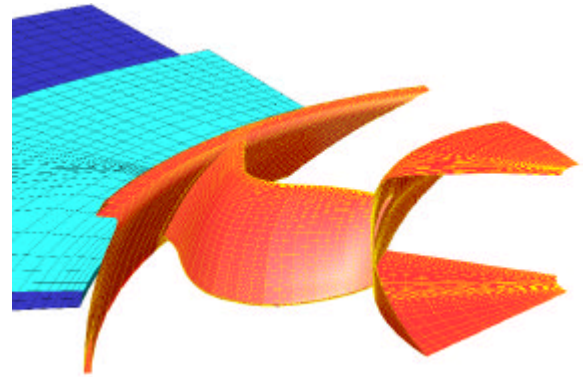


Figure 2: A close-up view of the fan hub, blade, shroud, and plate

Figure 2 shows a close-up view of the fan hub, blade, and rotating shroud. To emulate the conditions of the AMCA test chamber, the upstream and downstream plenum chambers were included in the CFD model. These have a diameter that is much greater in size than the fan. The stationary flat plate positioned radially outside the rotating shroud acts to separate the high and low pressure sides of the fan. The upper and lower surfaces of this plate are shown in Figures 1 and 2 in aqua and dark blue.

A contour plot of the computed static pressure distribution on the blade and rotating shroud surfaces is shown in Figure 3. The view is from the upstream side of the fan. The contours show low pressure



Figure 3 : Pressure contours on the fan hub, blades, and shroud

(blue) on the suction side near the leading edges of each blade, with high pressure (red) on the pressure side just barely visible at the edge. (See insert.) High pressure is also evident on the inner faces of the rotating shroud, especially in the regions adjacent to the pressure sides of the blades.

Figure 4 shows path lines depicting details of the flow field between the hub and the rotating shroud. The path lines are displayed in the rotating frame. They are colored by pressure, and twisted by the level of turbulence intensity. As expected, low pressure is evident on the upstream side of the fan, and high pressure on the downstream side. Even though the twist looks fairly uniform, local fluctuations in the turbulence do exist, especially near the leading edge of the blades.

Numerous flow characteristics predicted by the FLUENT results were compared to experimental data. At peak efficiency, the static

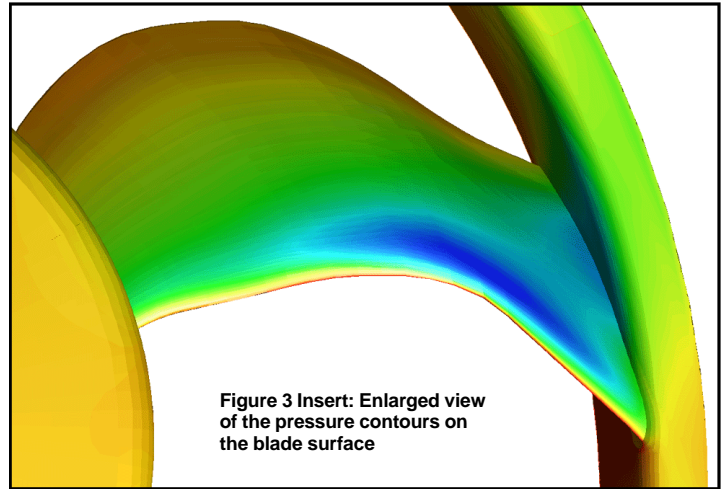


Figure 3 Insert: Enlarged view of the pressure contours on the blade surface

pressure rise across the fan was found to agree with the experimental measurements within the range of data uncertainty. At higher flow rates, room for improvement was indicated, and will be explored through modifications to the grid and turbulence modeling strategies in future simulations. Also quantified was the tip leakage of the flow. Flow separation and wash at the blade trailing edge, both of which have adverse effects on the fan performance, were found to be at an acceptable level for this model. Fan efficiency and the

distribution of turbulent kinetic energy were also of interest. The latter is important in evaluations of the level of fan noise.

Overall, the CFD analysis provided a wealth of data that would be hard to obtain otherwise. This fact, when combined with the demonstrated agreement with experimental data, suggests that the solution methodology used is an excellent choice for fans and similar types of rotating machinery.

Courtesy of Siemens VDO Automotive

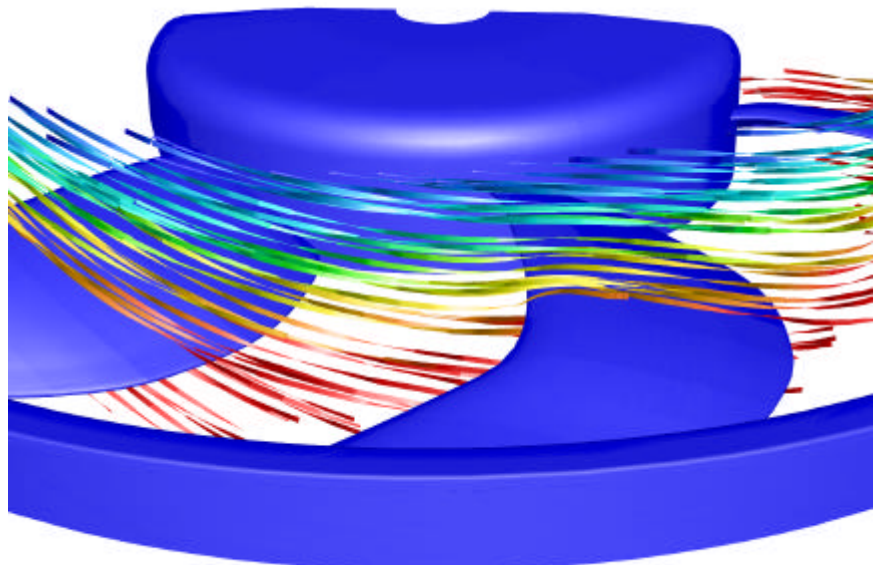


Figure 4 : Path lines colored by pressure and twisted by turbulence intensity show the flow through the fan in the rotating frame