A SIMPLIFIED CFD MODEL FOR THE RADIAL BLOWER

Farzam Roknaldin
Applied Thermal Technologies Inc,
3255 Kifer Road, Santa Clara, CA 95051
Phone: (408)-522-8730 ext. 208
Email: roknaldi@applied.fluent.com
Ridvan Amir Sahan (1) Xiaohua Howard Sun (2)
Fluent Incorporated
3255 Kifer Road, Santa Clara, CA 95051
Email: ras@fluent.com (1), xs@fluent.com (2)

ABSTRACT

Detailed level Computational Fluid Dynamics (CFD) models for fans and radial blowers involve information about blades' geometry, flow angles, blade's rotational speed, and flow approach velocities. Accurate simulations of such models require large numbers of mesh points which is beyond the allocated time and available resources for engineering design cycles. When dealing with system or board level thermal analysis, where a fan or a blower is among many components need to be modeled, a "macro" representation of a fan or a blower is preferred. A "macro" model for a fan is a plane surface that induces pressure across as the flow passes through it. The pressure-airflow relationship is taken from the fan curve provided by the fan manufacturer. A "macro" model for a radial blower is more involved because of the 90° flow turn inside the blower's housing and induced flow swirl caused by impeller blades. The need to capture the flow turn and induced swirl becomes more pronounced when simulating multiple interacting blowers inside a blower tray. In this paper, a systematic approach is presented to design the blower macro from the existing fan model. The Icepak CFD results for the blower tray have been analyzed and compared with the experiments conducted at Applied Thermal Technologies Laboratory. A typical use of a three-fan blower tray in a system representing telecommunication application is also presented at the end.

KEY WORDS: CFD, Icepak, Radial Blower, Fan Curve, Thermal Analysis, Macro Model, System Level Modeling, Experimental Measurements.

MODELING CONSIDERATION

In this paper, the main emphasis is to build a simplified blower model from an existing fan model, which is readily available in typical commercial CFD softwares. Icepak (1), Fluent's flow and thermal analysis software, is used as the primary tool for numerical modeling of radial blowers in a system level design. Although we have explored ideas and features built in Icepak, the methodology that is proposed here can be implemented and verified regardless of the software tool used.

RADIAL BLOWER DESIGN

Figure 1 illustrates the blower geometry built from Icepak primitive objects such as solid, hollow, fluid blocks and fans. Blower's housing, inlet and exhaust areas can be seen from this figure. There are two fans built into the blower model. Figure 2 shows the position of these fans relative to each other. The inlet fan (smaller of the two) is mainly to induce pressure and part of the swirl that is actually induced by rotating backward impellers. A corrected blower curve, as will be discussed in the next section, is assigned to this fan. Second fan (larger of the two) is merely used to enhance the swirl, therefore, do not increase pressure across it. Special consideration is given to capture the swirl mainly because of interaction of multiple blowers in a single tray.

The value of the swirl is adjusted in such a way that the flow can exit blower at 45° angle as seen by experiment. Here, no specific study has been done to formulate the swirl, leaving it open to exploration in a future work.

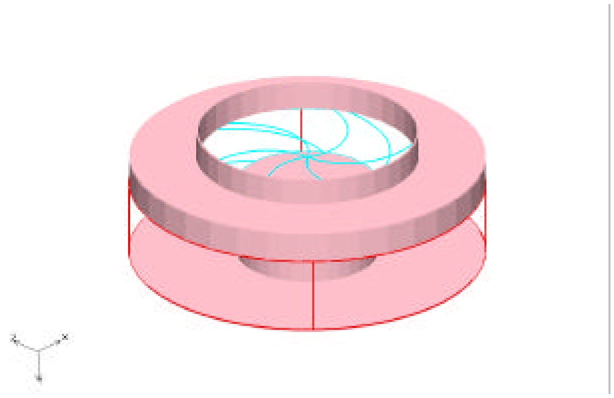

Fig. 1 Blower's solid model

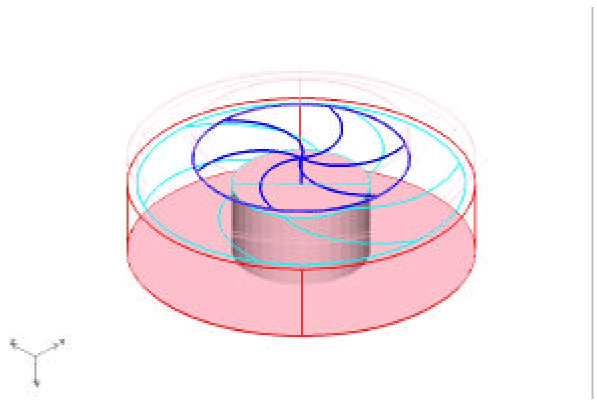

Fig. 2 Fan locations relative to each other

CORRECTED BLOWER CURVE

Blower curve provided by the manufacturer needs to be modified when used in the proposed model for the system level modeling of electronics cooling applications (2). Modification is required because of the fact that the blower, when compared to the fan, includes that extra pressure required for the 90° flow turn from inlet to exhaust. This extra pressure needs to be added to the fan curve to account for the flow turn. This pressure difference is mainly described by inviscid Bernoulli equation, which relates pressure drop, DeltaP, to change in momentum. From this, it can be shown that the relationship between the pressure difference and volumetric discharge, Q, passing through the fan is quadratic (i.e. DeltaP=cQ^2). The correction constant, c, can be readily obtained by a single simulation.

Below summarizes the steps need to be followed in order to correct the blower curve:

a) Build the single-fan blower geometry from Icepak primitive objects as described earlier. Associate the manufacturing blower curve to the inlet fan. This curve is shown in Figure 3.
b) Make the computational domain the same size or slightly larger than the blower. This ensures no other pressure drop other than the one associated with 90° flow turn.
c) Run the simulation to compute total CFM passing through blower. Ideally, one should obtain maximum CFM indicated by the curve. However, CFM is lower due to 90° pressure drop.
d) From the blower curve find the pressure corresponding to the computed CFM. This is the maximum pressure loss due to the turn.
e) Construct a quadratic blower resistance curve, DeltaP=cQ^2, where the two end points are the origin and the maximum pressure loss point as obtained in (d) (see figure 4).
f) Modify the pressure side of the blower curve by adding the curve obtained in (e) to it (see Figure 5).

Figures 3 and 5 show typical and modified blower curves. Due to the quadratic nature of the correction, modified curve never vanishes at the maximum CFM. However, it can be forced to vanish by adding extra point (as shown in Figure 5). This should not cause any problem since no blower operates at the maximum CFM when placed into an actual system. By comparing Figures 3 and 4, it is interesting to notice that almost 17% of the blower's CFM is neglected if one does not properly modify the blower curve.

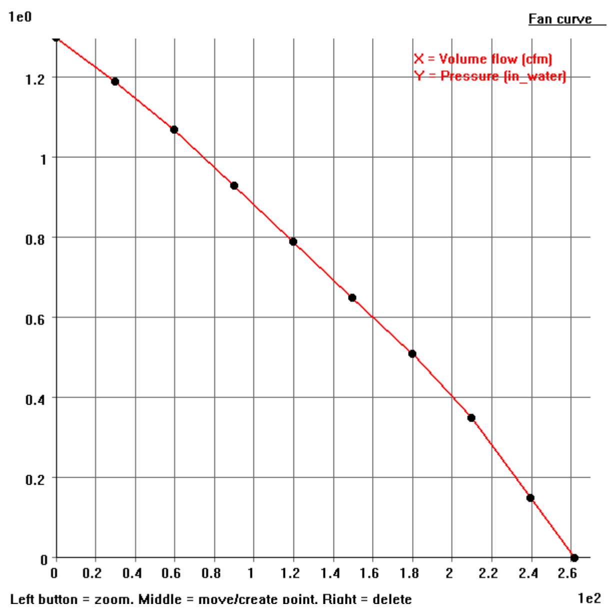

Fig. 3 Typical blower curve

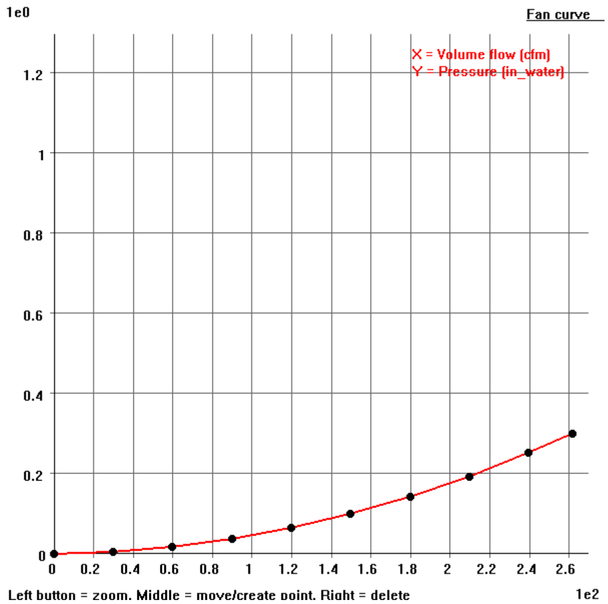

Fig. 4 Blower housing resistance curve

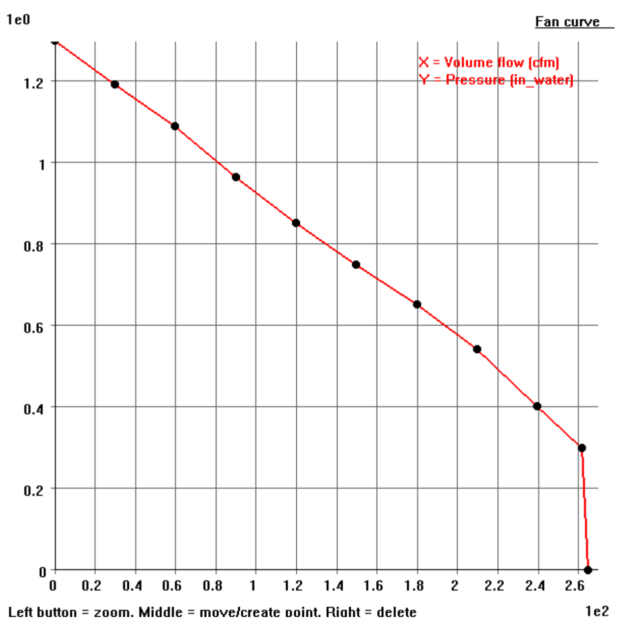

Fig. 5 Modified blower curve

THE BLOWER TRAY

Proposed blower “macro” model can be used to create a blower tray. Figure 6 illustrates a tray containing 3 ebm R2G175 blowers. Figure 7 shows the actual blower tray, oriented upside down, which is tested at Applied Thermal Technologies Laboratory. Using the wind tunnel, system airflows at different backpressures were measured. Backpressure is an additional pressure representing a system attached to the blower tray. Wind tunnel test set up is shown in Figure 8. Table 1 shows the comparison between the simulated values of CFM and the measured ones at different inlet backpressures. As can be seen, the agreement is fairly good, however, the simulated values are slightly higher. In reality, there are pressure losses that are not considered in our CFD model. An important loss here is the flow interaction between blower blades that changes the blade aerodynamics, hence, degrades the blower curve. Measuring the individual blower curve is beyond the scope of this work. A common fix is to de-rate the curve by certain percentage verified by experimenting. However, for the consistency purpose, we did not de-rate the blower curve. Figure 9 shows the velocity distribution within the fan tray obtained from a simulation with no backpressure. As can be seen, the swirl pattern has been captured very well. Figure 10 shows the speed contours in a plane cut that corresponds to the measured velocity data (shown in Figure 11). Notice that exhaust airspeeds are very non-uniform and have their peaks at either edge of the tray exhaust. Table 2 shows the velocity comparison, at different points in the exhaust vent. As can be seen values are within 10% of each other.

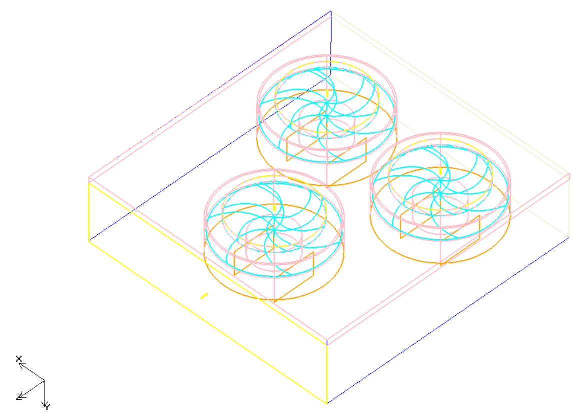

Fig. 6 Modeled blower tray with 3 ebm R2G175 blowers

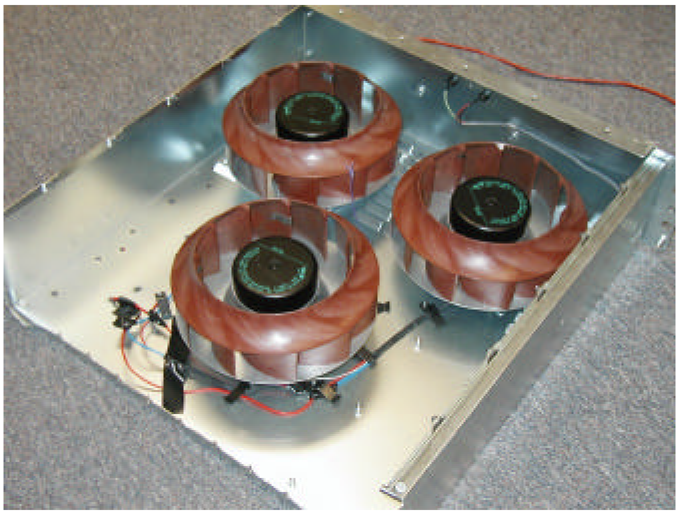

Fig. 7 Actual blower tray of 3 ebm R2G175 blowers

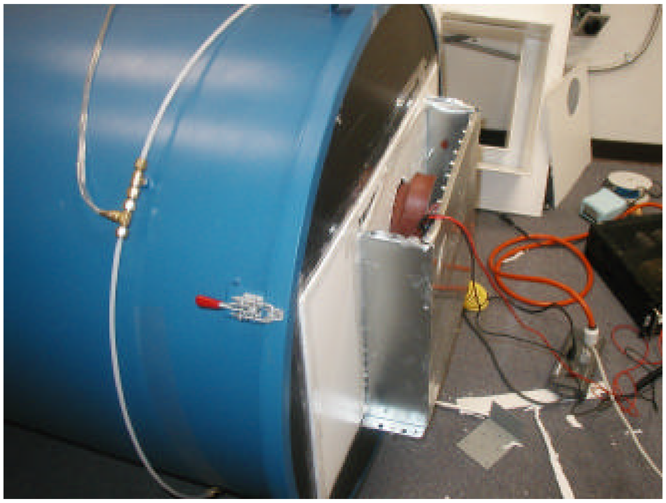

Fig. 8 Wind tunnel test set up to measure airflows at different backpressures.

Values of back pressure	P=0 (in_water)	P=0.1 (in_water)	P=0.2 (in_water)
Experiment (CFM)	630	580	520
Computation (CFM)	645	600	550

Table 1 Measured versus computed values of CFM at different inlet pressures

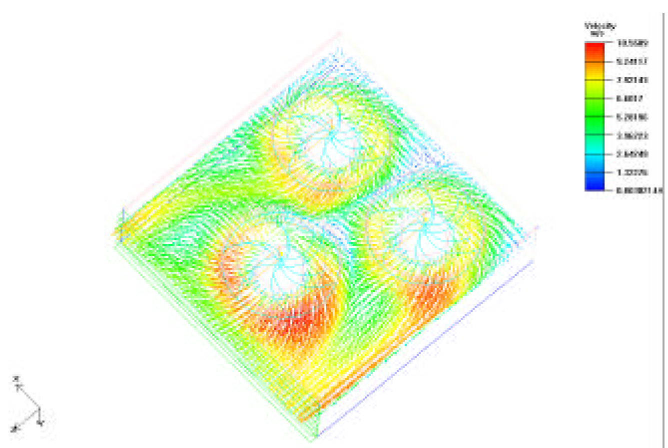

Fig. 9 Velocity distribution within the blower tray

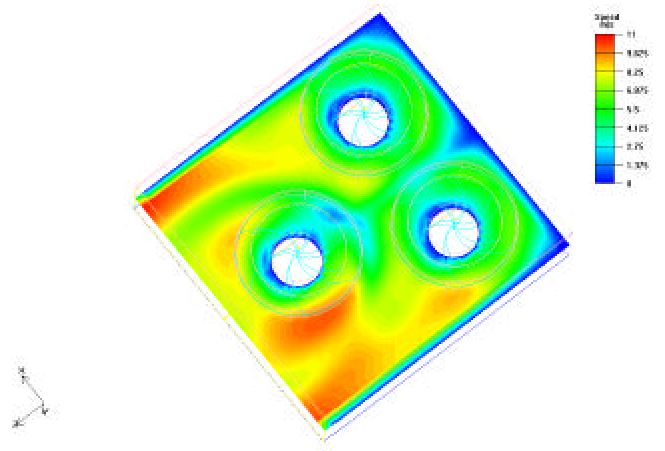

Fig. 10 Speed contours in a cut plane where experimental data were collected

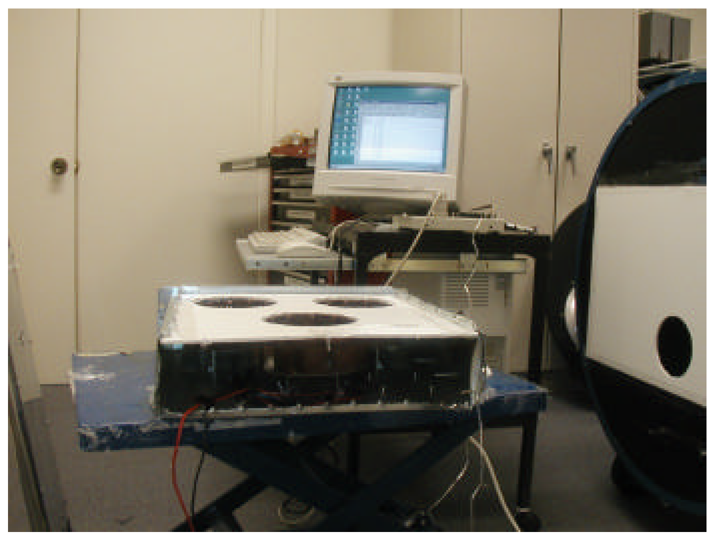

Fig.11 Measurement of local velocities at exhaust

Velocity	Left edge	Mid left	Middle	Mid right	Right edge
Experiment (m/s)	10.5	9.5	7.5	7	10
Computation (m/s)	10.1	8.8	7.9	7.7	10.3

Table 2 Simulated versus measured velocity distribution at blower tray's exhaust vent

BLOWER TRAY IN THE SYSTEM

At this stage, the blower tray is ready to be used inside an actual system for thermal design purpose. Figure 12 illustrates a typical telecommunication system when blower tray is used for cooling purpose. Figure 13 shows the velocity distribution on the three-blower fan tray. Figure 14 shows the velocity distribution between the cards and passing through one of the blower fans and leaving the system.

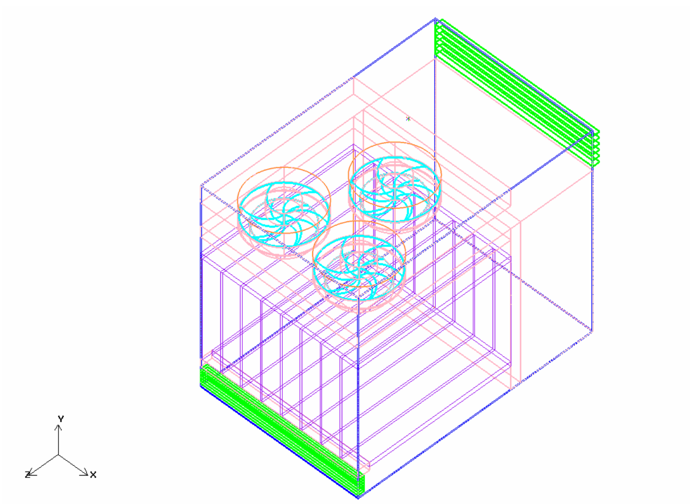

Fig. 12 Blower tray on top of the system

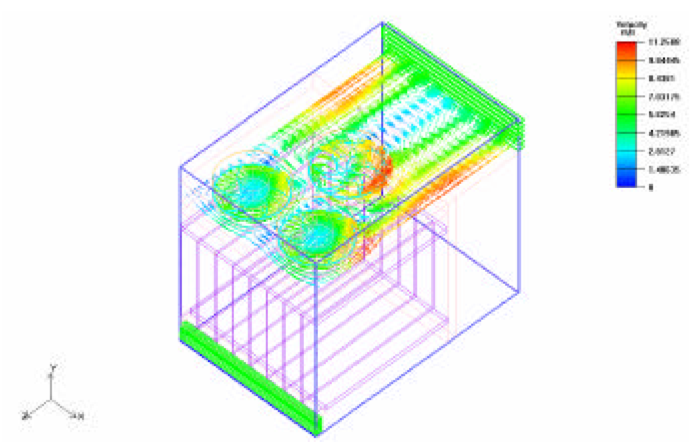

Fig. 13 Velocities and swirl within the blower tray

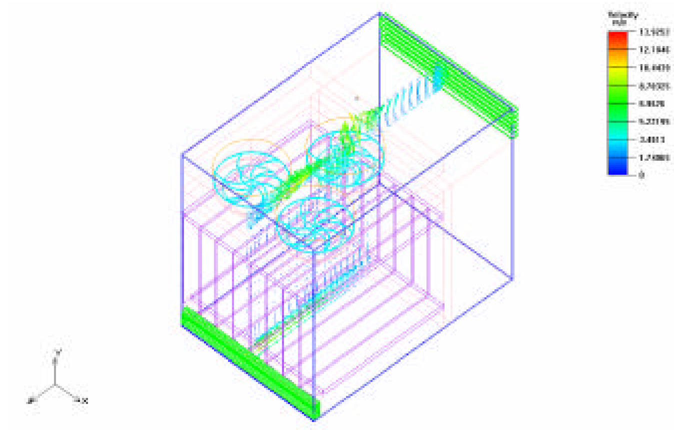

Fig. 14 Velocity distribution in a plane cut between two boards

CONCLUSION

In this work, a systematic approach has been used to model blowers/blower trays that are widely used in system level thermal design. Detail level model of impellers in CFD simulation requires large number of mesh points and is not yet feasible for engineering design cycles. Using simplified or "macro" approach, proposed to model blower with corrected fan curve, one can produce accurate and fast results. Blowers are integral part of the cooling methodology in most of those systems where the power densities are quite high. CFD tools such as Icepak greatly reduces the pre-design effort concerning prototyping and experimenting. The design approach used in this study may enable the thermal analyst to correctly model the cooling mode of an electronic system on a timely fashion.

Acknowledgements

We would like to thank Applied Thermal Technologies Laboratory for performing the experimental measurements.

References

[1] "Icepak Version 3 Users Guide", Fluent Inc., Lebanon, New Hampshire, June, 1999.

[2] Yunus A. Cengel, Heat Transfer (A Practical Approach), McGraw-Hill, 1998.